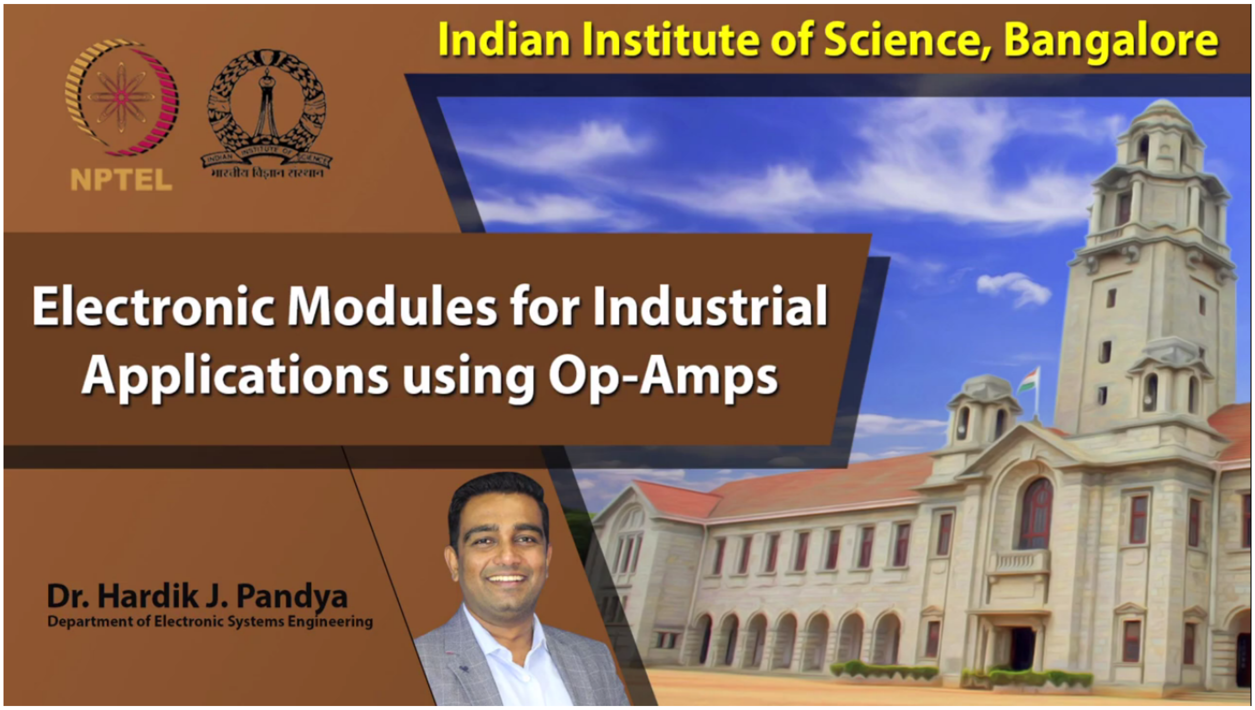
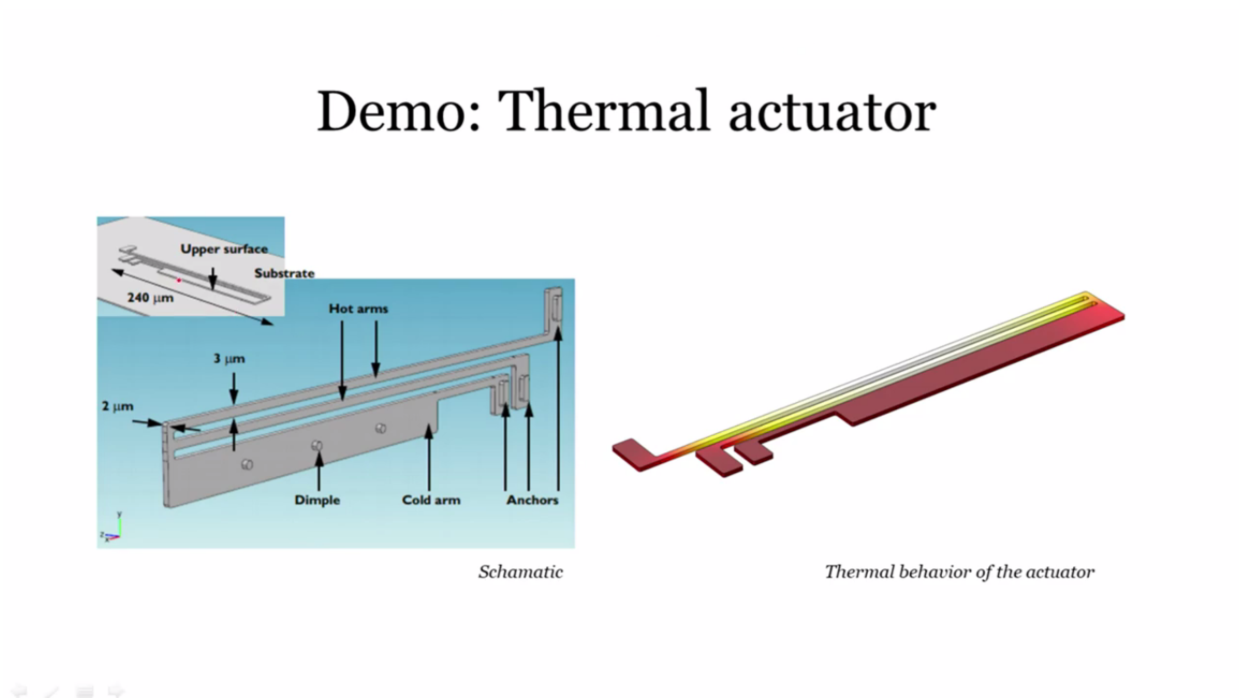
Indian Institute of Science

भारतीय विज्ञान संस्थान

NPTEL

National Programme on Technology Enhanced Learning

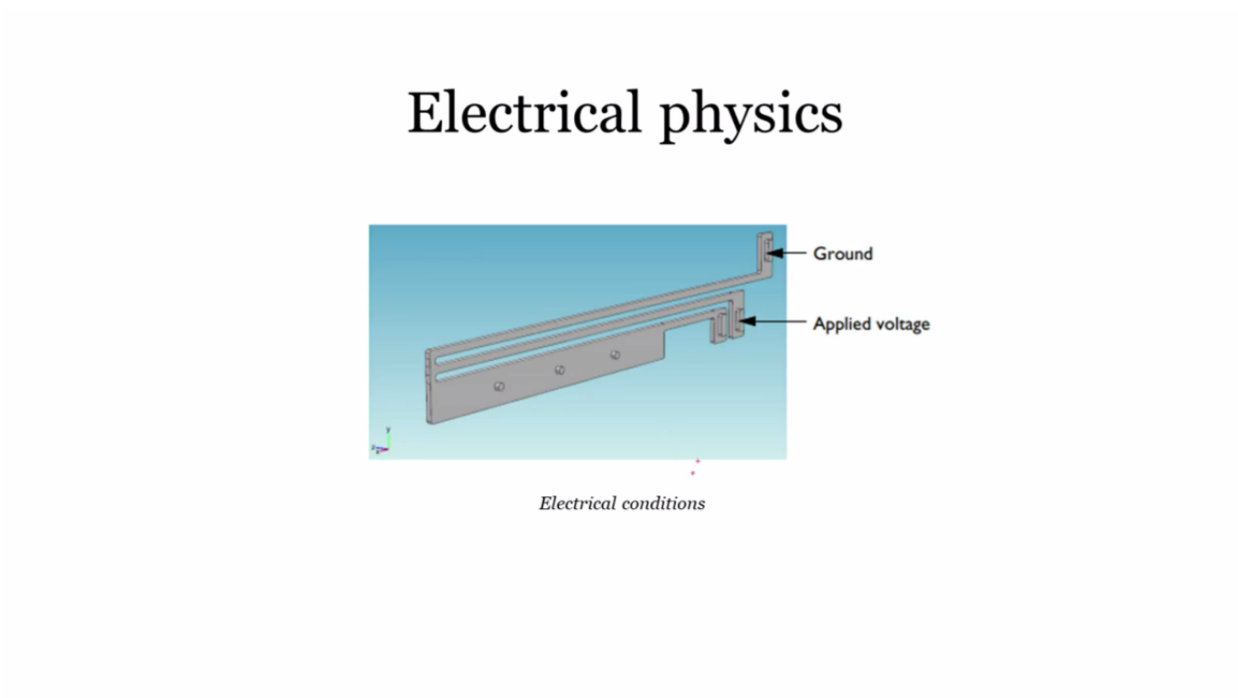




So let us continue with the demonstration of how you can actually do thermal actuator. The thermal actuator as you can see over here in the screen you have one polysilicon which is having the length in order of micrometers and you can see over here it's having the thickness of around only 2 micrometers. And the leg that you can see over here, the leg which is going to get heated up which is also known as the hot arm is having the width of around 3 micrometers. There would be three anchors with which it's going to get attached to and there are three dimples with which it's going to roll on with.

The main application of this thermal actuator is to give a motion. For example, you want to give an input of a voltage and you want to see if this particular system moves with the particular value in order of micrometers. This system being in order of having the length in order of I think it will in order of 100s of micrometers will move in order of within tens of micrometers. So let's see how to actually model this kind of a multi physics coupling because over here there are three physics that are going to get into the picture. The first physics is the input current that we give.

Because of the input current or the voltage that we give there would be a certain rise in temperature because of joule losses and because of the joule losses there is going to be some kind of thermal expansion because each material has its own thermal expansion coefficient. This is what we are going to couple. We want to couple the electrical physics with the thermal physics and the structural physics which leads to deformation. Our end goal is to see if we change the voltage how much is the maximum displacement does the thermal actuator gives.



So let's go and try to understand the physics step by step. The first physics is the electrical physics. So in this circuit that you can see over here what you see over here we actually are going to apply a particular voltage at one of the edge over here that is one of the anchors over here. And we want to pull the ground some where over here. The voltage with which we give could be actually coming up through a very big circuit.

Let us now go to COMSOL. So this is the first screen that you will see once you open COMSOL. If you are little bit experienced you can go ahead and make a blank model. So you don't need to add any physics. Just one click will lead you to a complete new model, a blank completely new model. The Model Wizard is a step-by-step process to make or initialize your model.

So let us click on Model Wizard. And it's asking us for the space dimension whether you want to make a 3-dimensional object, 2D Axisymmetric, 1D Axisymmetric, 1D or 0D. So let us go ahead with 3D. Now it's asking about which physics that you want to add. As of now we know that there are going to be three physics that are going to play role. The first physics is the electrical physics. Second is the thermal physics and the third is the structural mechanic physics which is going to lead for deformation.

The first thing that we are going to add is only the electric physics. We will see how the potential is varying throughout the domain. How is the current getting accumulated at some part of your domain which is actully getting lead – which is actually leading to the thermal losses and that losses is eventually going to go to the thermal step. So our first step is to do only the electrical physics. So let us go into the AC/DC module. So this is the AC/DC part. And in this AC/DC part there are a lot of different different interfaces. Electrical current interface, magnetic field interface, rotating machinery magenetic interface where you can model motors. As of now we will go ahead with only the electrical currents. This specific interface is to model current through conductors. If you are talking about capacitors then you need electrostatics. So let us go with electric currents and add.

So I can go to the next part as the study. Here there are many different kinds of studies we can perform. We can do frequency domain study which allows us to do harmonic input. We have stationary study where we can give steady state analysis that is we give only a particular voltage. Then we have time dependence study which helps you to give you transient analysis of your complete system. So all these three different kinds of study can be performed.

Along with this we have small signal analysis wherein we give a particular DC bios along with the AC component. So let us go with the stationary analysis and click on Done. So the approach that you can follow in the COMSOL is the top to bottom approach. That is you first create the geometry, add the material, add the physics and then perform the machine. Then you do which kind of analysis you want to perform and finally the results. So this is the top to bottom approach that usually is prefered in COMSOL. One more approach preferred is in the ribbon pane that you can see in the top. Here you can see that the same thing that you have already seen in the model builder. You have to first create the geometry, add the materials, apply the physics, perform the machine, analyze which kind of analysis you want to perform, then finally the results that is to analyze your result.

We also have one more block as developer that is if you want to write your own code along with the code that being solved by COMSOL. So the first thing is to make the geometry. However, to save the time I am going to import the geometry. There are two ways to import the geometry. One is to import through some other tool any other CAT tool and if you want to insert a sequence. A sequence is like a step-by-step process to make the geometry. If you want to insert that sequence from a pre-built COMSOL file you can use a Insert Sequence. So let me choose Insert Sequence. And I choose the model that I have already created. And I click on zoom extends.

So this is the geometry that it's already available in the libarary file. So if you want to go through this example model which is also known thermal actuator, you can go to File, Application Library, and just search for thermal actuator. And you will see the thermal actuator model file available over here. So I have added the geometric sequence as you can see over here. The next thing is to add the materials. So let me just add the material by right clicking the Material, Add Material from Library. I can go ahead in the built-in. I can have the polysilicon material property.

Now we can see that over here the material property it is including electric connectivity which is being defined as sigma as a function of T. This T is the temperature. What it means is that the sigma is not constant. It is a function of temperature. If you want to see what is the relationship you can open this polysilicon and then go into the analytical function we just define the conductivity and if you can plot you can see how the conductivity is decreasing with the increase in the temperature.

The next thing is to give the physics boundary condition. As you can see over here we give apply a particular voltage at this anchor and ground at this particular boundary. There are different ways of applying the boundary. For example you can use electric potential. One another way is to use terminal if you want to evaluate the resistance offered by your system. For example, I want to know the resistance offered by a system. So I will go ahead with the terminal and I apply that particular boundary over here and I give the terminal type as voltage and I choose the voltage as V0. So right now you can see that it is showing unknown variable because we have given a variable name but we have not defined the parameters. So let me go to the parameters. This all parameters have already been assigned or it has already come up because we have imported the sequence of geometry steps. So these all parameters are defining the geometry steps. Let me define now V0 which is around 5 volts. That is the operating voltage.

So now the error has ceased to existence. So there is no error. We have given a voltage over here. So let me also give a ground boundary condition on this particular part over here. Okay. let me go to the mesh and click on Build All mesh. So this mesh that you can see over here is fine enough. If you want to improve the mesh you can directly go to Fine or Finer with me I think fine should be good enough. Now, you can see that you are actually the dimple as cylindrical. But over here it's not even a cylinder. So this mesh might not be sufficient enough because your geometry might be having the cylinder. But the physics is going to be solved only how the mesh is being – so it's actually going to solve the way the mesh behaves or looks like. So we need to go ahead and also increase the machine. Let me go to finer. And now you can see that the mesh has nicely resolved the cylinder.

So now this is what looks good to proceed further. So let me go to my Study 1. let me write Electrical study. So my Study 1 is electrical study where I am only performing the electrical analysis. Let me go to compute and click on Compute. So now you can see the voltage distribution or the potential distribution along the polysilicon. Now if you want to see the current distribution the current profile. So there are many ways to see. The first way is to know how the current is flowing through. One of the way to do that would be to right click on the Results, 3D Plot Group. In the 3D plot group I can create an arrow volume. In the arrow volume I can search for the current. So in the replace expression I can search for the current density and click on Plot.

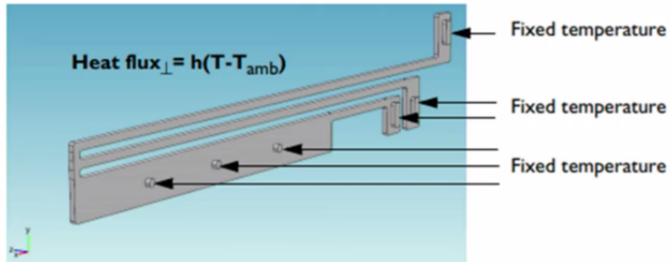
Still now I am not able to visualize the current properly. Ideally the voltage which shows that it's very high over here and it's decreasing like this so our current profile should also be similar to the gradient of your potential. The better way to recognize it would be to change the grid points.

So let me just go ahead and change the grid points. Hopefully we will be able to see those. So in X and Y I need to increase the grid points and in the Z point let me use only one. And now you indeed are able to see the current flow. So now we can see that the potential which was applied very high over here is actually making sense to the current is actually flowing like this. And it's going to turn over here. And the it's going to move back towards the minus X direction to the ground [00:15:20]. So this actually makes sense, the results makes sense. So it's always good that once you actually get the results you also validate the current profile, how much voltage are you getting to revalidate whatever input you are giving is actually being conceived by your system.

So the next thing is to understand there is a current flow. And we need to understand where the current flow is going to be very high. To understand that what I will do I will create one 3D plot group and I can create one multislice plot. Multislice plot would be somewhere in the More Plots over here. And I can search for norm of current density. So I can just search about current density, over here. And now you can see that the current density is really high somewhere over here that is in the thin like over here and over here. This means that the current at this particular, the current density at this particular legs is very high as compared to the other domains. So that also means that the losses which is proportional to the current are also going to be higher at this particular legs. So let us also see now if there are losses because of the joule losses how I can evaluate this. One way to see it would be again the multislice plot. So I can create the 3D plot group by right clicking on the results. And I can create one more multislice plot. And I search for losses. I have volumetric loss density and that is complete electromagnetic and then only the electric part. There is no magnetic component so we can choose any one of them. Let me choose Qh and click on Plot. And indeed that's what it is showing me. The losses are very high at both of the legs. You can see over here thin legs. It is even high at the corners and that's the same thing what the current density was showing that whenever the current density is high the same place the losses are going to be very high. So these losses are as you can see in order of watt per meter cube. Now you still have the losses but you don't know the losses are going to lead to watt change in the temperature. This system is working with which ambient temperature you don't know that because we have not assigned it yet.

So let us also set up a thermal condition to see how the temperature rises if we give a voltage of five volts in room temperature. Before that what I will do to – so because we have many studies that we are going to have to reduce the complexity I will group my results. So I select all of them and then I right click on the results on the selected notes and I click on Group. So my Group 1 is Electrical analysis.

Thermal conditions



Thermal conditions

The next part is I am going to add the thermal physics. So I go to physics, Add Physics. And I add now in Heat Transfer, Heat Transfer in Solids. Now you can see that once I again go back to my polysilicon earlier the only electric connectivity and the permittivity were having a green tick mark. But right now as we have added the thermal physics the heat capacity at constant pressure, the density as well as the thermal connectivity has been activated. Again you can see that the thermal connectivity is a function of temperature. The T now is actually coming from here once we do the coupling. Even the conductivity, the electric conductivity as a function of T, the T would be coming from the heat transfer in solids. In this let me just go over here the thermal conditions and as you can see over here we would be giving fixed temperature to the places where we have attached it to some other boundaries. For example, the dimples over here, this one, this one and this one which are on the cold arm and the anchors which are this one, this one and this one. And all the others we are going to give a convectic heat flux with a heat transfer coefficient of value 5.
So let me go to physics and now over here let me add the fixed temperature. To add the fixed temperature I will right click on heat transfer in solid, I go to temperature and over here I click on these boundaries and by default this is the room temperature. So as well as the top of the dimple. I click those. So these boundaries are going to be attached or it is going to be tied up with a particular temperature and it cannot increase from that.

Now we are also going to add the convicted heat flux. To do that just right click on the heat transfer in solids and you can use the heat flux boundary condition. So choose those. And I just click on my graphics, left click and then I use Ctrl+A. That selects all the boundaries and now I want to deselect the boundaries of the anchors and the dimples.

The next thing are we giving inward heat flux? No. are we giving convective heat flux? Yes. So I enable this and I also need to give an external temperature which is a room temperature by default. That looks fine. And the heat transfer coefficient as five which means that there is a air flow, not even air flow actually there is a air surrounding this particular system. If there is air flow the thermal expansion – the heat transfer coefficient will increase. If there is water which is cooling the system it will even increase even faster. So this kind of heat transfer coefficient actually depends upon what is the ambient temperature or what is the cooling condition that you are trying to give. For example you can also give over here external natural convection, internal natural convection, external force convection, and internal force convection. These are all the options that are available.

So we are done with the physics with the thermal physics. If you want to go back to solid one you can see the actual equations that is going is actually being solved over here. Similarly in the current conservation node over here you can see the actual equation that is being solved by the electric physics.

Now coming to the coupling part. How the electrical physics and the heat transfer physics are getting coupled. That if you would have been careful enough you would have seen that these multi physics node was not coming up before. But right now it has come up because there were two physics that could be coupled. There are many automatic coupling in addition to that there are also manual coupling. So if you want to add the automatic coupling just right click on the Multiphysics node and you can see over here the Electromagnetic Heating.

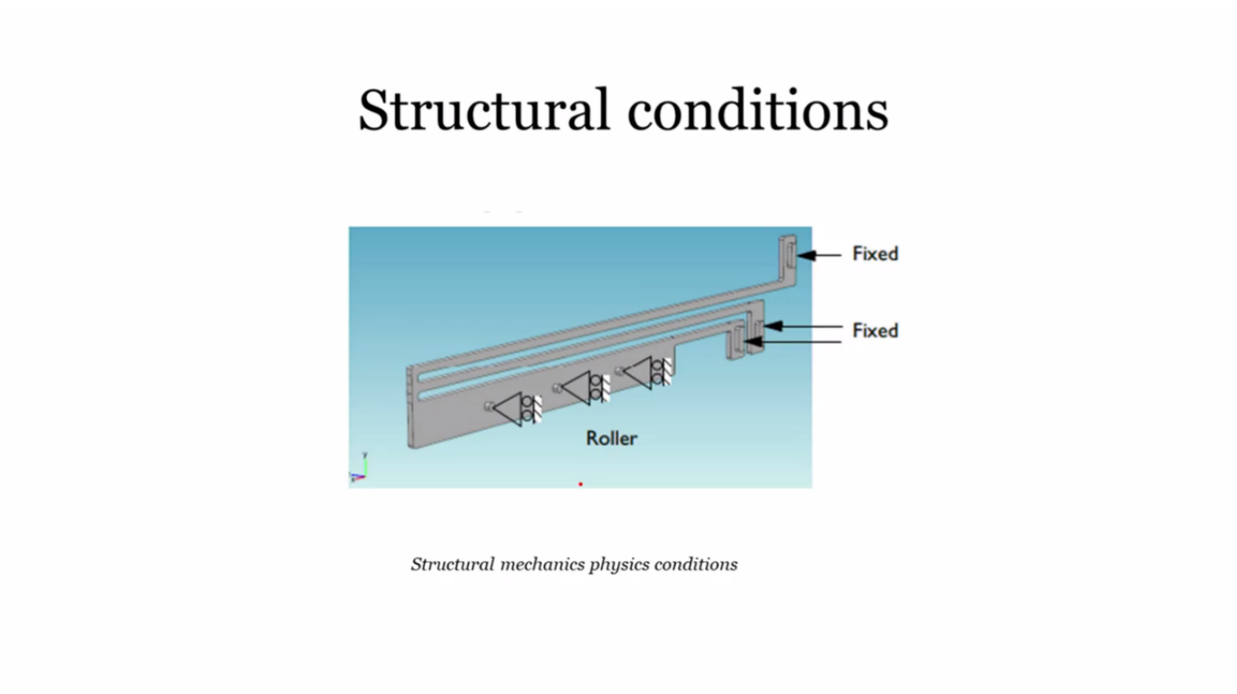
So let's click on the Electromagnetic Heating. This as you can see it couples your domains and the boundaries from the electrical currents to your heat transfer in solid interface. So whatever losses that were getting accumulated by your current that is going to flow all the joule losses are going to get transferred into the heat transfer in solid part. And that would be responsible to the rise in the temperature. So now let me go to the mesh. Okay I need to add one more multi physics coupling that is temperature coupling. And let me go to the mesh. There is no change in the mesh anyways. So we can now go ahead and add one more study. So I go add one more study. And over here I add one more stationary study. Over here I write electrical and thermal analysis because we are going to do both now and you can see that both have been activated over here with the multiphysics node.

It is very important to note that your results would be accurate enough only if you match your material properties as much to your real properties. If it deviates from your actual experimented property then the results also will deviate. So it's very important to come back and see to the materials that you are using because this is very very important.

So let me go to my study 2 and click on Compute. You can also see the conversions plot with the error. You can also manually tweak the amount of error that you want to use that is related tolerance error that you want to use to make the system converge. So now you can see the first plot is related potential, the second plot is the temperature and as we know we also have seen before that the temperature rise would be somewhere over here because if you see the electrical analysis in the losses plot the power losses plot this is the power losses plot that you can see over here. There are a lot of heat losses over here. So this is the temperature plot. Now for example you want to see the maximum and minimum values over here you can see over here. You want to see the actual values without the exponential part. You can actually come to number format. And you can use the precision values. You can increase that. So like you can see that it's very very high temperature.

And it's also important to note that COMSOL is a tool which will help you to give the results but it will not help you to tell that if it is going to melt or not. This is a part that we have to do by ourselves. For example if the melting point of our system is around 2000 kelvin so COMSOL will not tell you that it has like the temperature has rose beyond 2000 and it will give an error. So it will not give an error it will tell you that it is still increasing the temperature but you have to yourself understand that it's gone more than the melting point and this is how it actually stops.

Now for example you know the temperature and you want to now see the temperature plot from this point to this point in the line segment. How is this special distribution? To do that I can create a 1D plot group. So I right click on the results I create 1D plot group. And over here I right click on the 1D plot group and create a line graph. In this line graph I select which line I want to use so let me use this line. And I want to evaluate my temperature. One way is to directly enter the expression over here. One alternate way is to just click on Replace expression and search for temperature. Before we do it we have to be sure that we are using the data set of study 2. So let me go ahead and choose my data set as study 2 that is electrical and thermal analysis and then search for the temperature. So we can see that temperature profile variable has arrived over here. So I will double click on that. and whether I want it in kelvin or I want it in degree Celsius I can use that and click on Plot. So now you can see the spacial variation of temperature along this particular line.



Okay. The next thing is to perform these structural analysis. Before doing the structural analysis what I will do is again group my electrical and thermal analysis together and I can write over here electrical and thermal analysis. The next part is to perform electrical thermal and structural analysis. So let me go to my presentation and see how are the structural conditions. You can see over here there are fixed boundary conditions over here, here and here that are on the anchors. And on the dimples we have roller boundary condition over here, here and here. That means that these boundaries can actually roll. You can give a particular translation motion but you cannot give an up and down motion to the roller. However, the fixed constant are not allowed to move in any direction.

So let us go to the COMSOL and add these physics. So we already added the electrical physics and heat transfer physics. The next thing I go to the physics, add physics and I add the heat transfer in the solid. I just double click on this and over here, sorry I added the wrong physics. I need to add the structural mechanics. So I add I go to the physics I go to the Add Physics and I go to the structural mechanics and I add solid mechanics physics. Again we can go to the material section and you can see now the young's modulus has been enabled over here. The Poisson's ratio has been enabled over here. Both are a scalar value. If you want you can also make it as a function of temperature conductivity by just writing over here. You can make it a temperature dependent material property by just writing over here as for example 300 divided by T for example. So this is just for example. Let us go with the constant value. In my solid mechanics as you have seen over here we are going to give roller boundary condition at the dimples and fixed boundary condition at the anchors. So let me right click on Solid Mechanics and I give Roller boundary condition. You can see the equation. So all the boundary conditions the domain conditions would be giving the equation. So from the equation also we can understand how the motion is going to be. It means normal motion of your boundary is zero while it will allow the tangential motion.

Now we go ahead with the fixed constant. Again right click on the Solid Mechanics add the fixed constant part and choose these boundaries. You can see the equation that U is equal to zero. So both tangential normal or any angular displacement are not allowed. The next thing is again to use the coupling. So we have already established the coupling from electric currents to the heat transfer that is the joule losses leading to the rising temperature. Now we need to couple the rise in temperature to the structural mechanics using the thermal expansions. So because of the rise in temperature what is the thermal expansion. The way to add that would be right click on the multiphysics and then you click on the thermal expansion.

The other multiphysics node would not be of interest to you as of now. And then I add my domain. Once I add this you will see that okay thermal expansion coefficient is being added. That is also function of T. So you can go within it. You can see alpha over here. How it is the function of T. so we have added the thermal expansion. So we go to the study node and add one more study a stationary study in my stationary Study 3 I write Electrical thermal and structural analysis. And over here you can see that I have enabled all the three physics along with the three multiphysics node.

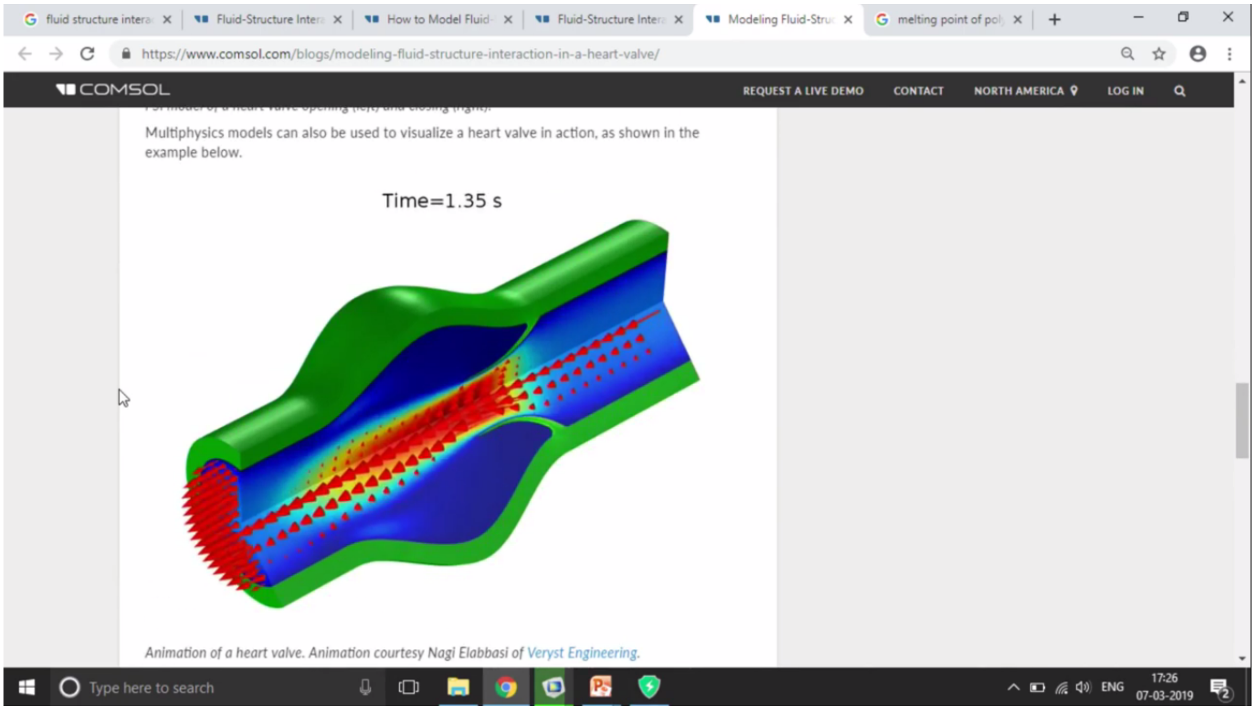
You can see over here the rest of the boundaries are free, by default all the boundaries are free but if you run your system stationary analysis that is study stationary analysis with the free boundary condition your problem becomes ill post and you will get some non-convergence issue. So it's always good to assign some fixed constraints boundary conditions. So let me go to my Study 3 and click on Compute. In the progress bar you can see the what [00:36:37] it is using how much is the iteration. We are not going to cover in detail the solver configurations. You can see the convergence plot over here.

If you have any questions please do not hesitate to ask your questions in the form of chat or the means that is allowed. Yeah so you can see the results now. You already have seen the temperature variation. Isothermal contours and the stress plot. So now you can see that the stress which are being developed with the surface plot and if you want to see it in Newton mm square you can just click on over here and search for Newton mm square or N/mm^2 or you can just directly mention over here Newton mm square and click on Plot.

So we will see the stresses over here. You can enable the maximum and minimum values so to see what are those values. If you want to see the displacement so in that case you can just right click on the results, 3D plot group. In this I create the volume. And in this I search for displacement. So you can see over here I mean we have to see the data set with which we are studying. So we are doing electrical thermal structure analysis. So let me choose this. And over here I search for displacement. So you can see this variable solid.disp that is total displacement. So let me choose this. And you can see that it shows around maximum. So maximum I can enable over here. Maximum displacement over on 21.2 micrometers somewhere over here.

Now if you want to see or visualize how much deformation is going to occur based upon the voltage that I have given I can use – I want to visualize how much displacement. So I can just right click on the volume 1 and click on Deformation. I can enable the scale factor as 1. So this is how much it's going to displace. So this length of this one if you want to see you can actually evaluate – you can measure actually by right clicking on the Geometry, go to the Measure using the geometric entity level as point and the length I can measure from one end to the other end. So it's around 26 micrometer. The length is not that big. Okay sorry, let me start from this point and this point. Yeah so length the total distance, the total length is around 200 micrometers. And that particular device is bending around 21 micrometers along the minus Y direction. So that's good enough.

Now however, I want to know that if change my potential how much is my displacement going to vary. In addition to it I also want to see the maximum temperature that is occurring within my system as I change my voltage because I know that my polysilicon would have been having some kind of a melting point. So let me just go to internet and search for melting point of polysilicon. So it's around 1400 degree Celsius. So now I know I have some kind of threshold value which with it not go increase the maximum temperature not increase that. So now what I can do is I can sweep my voltage. To do that I can just right click on the Study 3 and I can do a Parametric Sweep and over here I can add my voltage V0 and over here I can write the range. I can write – I can just click on the range. I can start with 1 volt. The step of 1 volt and a stop of 5 volts. So it's going to move from 1 volt to 5 volts. And then I click Compute.



So till the time it computes there were couple of blogs on fluid structure interaction that I will like to showcase you. This is one of the blog and so all the blogs are having very nice animations. So it's – like very nice to actually understand the physical intuition of the physics for example this is a water balloon during inflation what are the velocity magnitude, stresses that are being developed and all those things available. So this is a fluid structure interaction problem where the balloon is getting filled. Fluid structure interaction we had one more blog. This is for heart valve. So you can understand with this change in the flow of velocity how the valve is actually moving. So it requires moving mesh technique over here.

So such kind of fluid structure interaction modeling could also be performed. So over here you can see this what kind of solvers are we using. So we are segregatted solvers. There are fully coupled solver. Segreegated solver. These are the two basic kind of solvers. Fully coupled solver takes all the physics together. Segregatted solver takes it step by step. There are also two types of solvers that is stationary, sorry, it's direct and indirect. Direct is more robust. It takes lesser time but it takes a lot of memory. The itirative solver actually takes longer time but it actually takes less memory. So we have got some results. So we have now we can see that we have five potentials. So in the 3D plot group I can see for 1 volt we are getting, I need to go inside this, for 1 volt we are getting the displacement of 0.02. For 2 volts we are getting 1 micrometers, 3 micrometers, 8.23 and 2.15 like this. So now if you want to see – if you want to create a graph of if you change the voltage how much is the change in the displacement I can just right click on the results. I can create 1D plot group over here. I can create a point graph and I can choose this point. And I can ask it to plot for once I select the data set, a proper data set that is Study 3 I can ask him to plot the displacement so I search for displacement somewhere over here and I double click on this. And I just click on plot. So it would be automatic. The X axis would be parameter in voltage and Y axis is going to be my displacement.

So you can see that as the voltage is increasing the displacement is also increasing and it's non-linear curve that we want to see, that you all are seeing.

You can also enable the markers over here. So you can use asterisk mark with in data points that is at whatever points you have swept you will get the results.

Okay. Now the next point is to see also the temperature. So the temperature is maximum temperature. To do that what I can do is I can just right click on my Definitions. I can create a component coupling. And over here I can choose Maximum. So I choose the Maximum over here. Select the domain over here and to make this maxop1 that is the operator name, maxop1 available in the post proceesing I need to right click on my Study 3 and click on Update Solution. That's it and over here I need to create a 1D plot group again. So I right click on the results. I create 1D plot group. In this 1D plot group I go ahead, right click on it and create a Global plot. Here I will use the same operator maxop1 as a function of temperature. And I use the data set as my third data set and choosing all of them. So I will use All. My X axis again temperature, sorry voltage. And now you can see that temperature, max temperature is increasing again as voltage is increasing and I can add an asterisk marks with in data points. So if you assume that our melting point is around 1500 degree Celsius so let us make it degree Celsius so if it's around 1500 then your device would only work till 4 volts. If you go more than 4 volts then your device is going to melt for example.

Okay. So this is how you can actually do the analysis. One more thing that I wanted to show is if you want – so right now if you want to share this so let me just group this first. Now this is the electrical thermal and structural analysis. Now if you want to share this particular model for example you could be researcher or you could be a student or you could be a professor who is taking this course and you want to share this complete analysis with your colleauge. So if you are professor you would like to give it as an assignment for your students and they want to change some kind of a parameter and get the different kinds of results at different different operating range or you could be a R&D person and you have specialized in electrical thermal and structural analysis but you want to give it this kind of model to your design engineers who would be designing the system so they actually don't need to actually go through the complete model. They are only interested in the result. So they can actually vary the input parameters and finally get the results. They are not interested to se what is happening over here inside. Or you could be a student who have make this model and is going to give a thesis presentation to a professor and they are also interested, the professors who are attending your thesis presentation they are interested to actually see at different different operating voltages or operating conditions what are the different different conditions at that time in [00:50:18] they want to change it.

So one way is to create an app. So in the next couple of minutes I will just show you to conclude this session how to create an app. It's very simple to create an app. That's how we will show it. To do the app it's important to actually parameterize. So I can again go to my Range. So I will write start_V, step_V and stop_V. So these three parameters I have defined that is my start voltage, my step voltage and my stop voltage. I will click on Replace. Again it's saying unknown model parameter because I have not defined. So let me write over here as start_V as 1 volt. It's always good to write your whatever parameter along with the unit so that there are no inconclusive results. Step voltage is 1 volt. And then the stop voltage is 5 volt. So it starts with 1, have a step of 1 and stop of till 5 volts. Again it's always good to see that you have not – there is no mistake in spelling mistakes and all those things.

So I have defined the parameters. I know what plots I want to see. So for example I want to see my thermal plot over here. I want to see 1D plot group 14 and 15. Okay. So I go to my Application Builder. So only by one click I can go to my Application Builder. And over here I can just click on New Form. Once I click on New Form it's asking me all the parameters that I want to change for example if I want to change the geometry, all those parameters can also be given. As of now we will give the start voltage, step voltage, and the stop voltage. The actuator length could also be given for example. So if you want to vary the length and see how the temperature is varying, that also is possible. Then the next thing is the graphics; what do you want them to see. For example you want them to see the temperature. You also want to see the stresses or the displacement. You also want them to see the how the rise in temperature occurs with the voltage and the displacement with with the voltage.

On the right side you will see the graph or the form that is being build and then what buttons or what kind of analysis they want to perform. They want to do only electrical analysis, electrical and thermal or electrical thermal and structural. So as of now let me give electrical thermal and structural analysis and click on Add Selected and if you want you can also add all the study, the first and the second one. So and just click on OK. So this my first results over here I can just move and modify all of them and make it little bit large so it is easy to visualize.

So now the compute button somewhere over here. So here I can just write over here and I can change the text as electro-thermal or thermo-structural study. So I can keep somewhere over here. I can move my total length somewhere over here. So this is my voltage condition, this is my geometric conditions and then this is my compute button for which kind of analysis button I want to do. So I click on Test Application. So this is how your app would look like.

So they can – your professor, your student or your design engineers can actually go ahead and change the actuator length for example 300 micrometer and then just click on this compute. And where the next set of results would be coming up. They can also change this start voltage for example 5 and step of 1 to stop of 10 and then they can compute the study and see the change in the results. You can also add the parameters as material property and change the material property over here. So this particular could be actually shared with your students and they can actually – this could be a standalone kind of a thing and they can actually vary all these parameters and gain lot of understanding of the system by themselves.

Okay. So this is how the complete simulation of COMSOL is. I hope you enjoyed the session. Thank you for attending the session. If you have any questions please do not hesitate to ask us. We will try to respond it as soon as possible. Thank you. Have a good time.