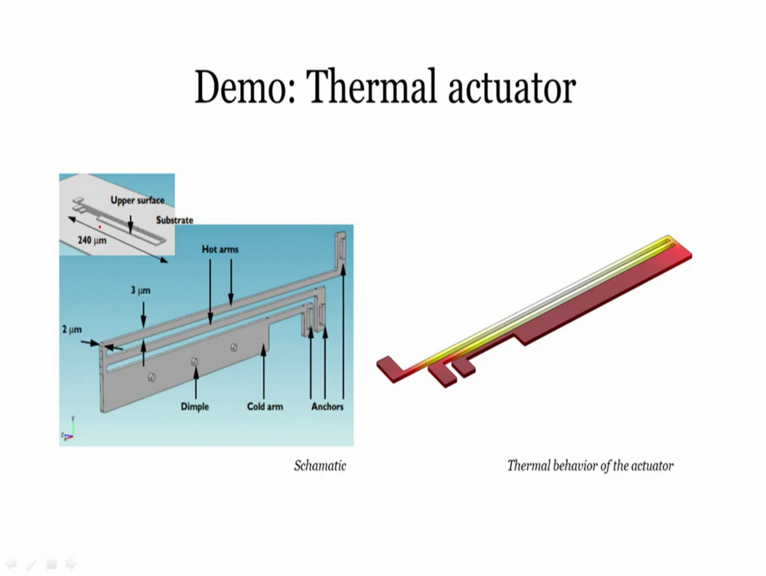
**Electronic Systems for Cancer Diagnosis**
**Dr. Hardik J. Pandya**
**Department of Electronic Systems Engineering**
**Indian Institute of Science, Bangalore**

**Lecture – 69**
**Demonstration of Thermal Actuator and Understanding of Application Builder**

So, let us continue with the Demonstration of how you can actually do a Thermal Actuator.

(Refer Slide Time: 00:37)

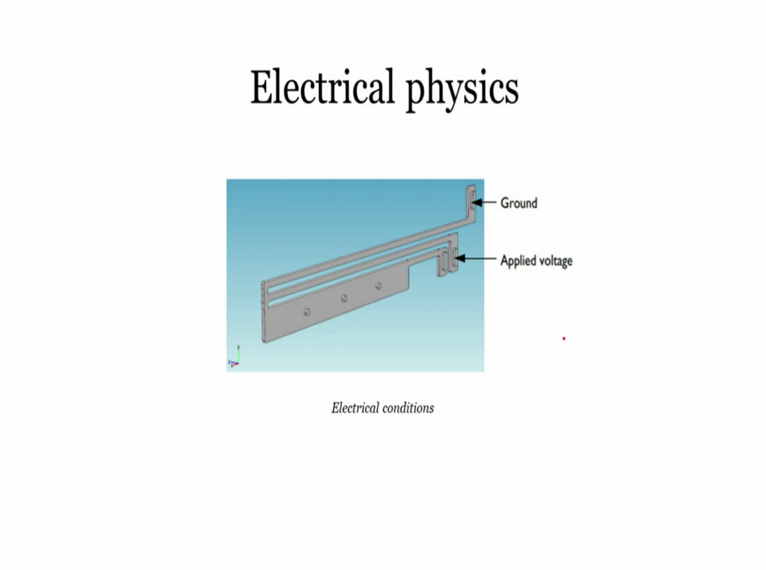



The thermal actuator as you can see over here in the screen you have one poly silicon which is having the length in order of micrometres and you can see over here it is having a thickness of around only 2 micrometres. And, the leg that you can see over here the leg which is going to get heated up which is also known as the hot arm is having the width of a round 3 micrometres. There would be 3 anchors with which it is going to get attached to, and there are 3 dimples with which it is 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 a particular value in order of micrometres. This system being in order of having the length in order of I think it would be in an order of hundreds of micrometres, would move in order of within tens of micrometres. So, let us see how to actually model this kind of a multi physics coupling because, over here there are 3 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 the Joule losses there is going to be some kind of thermal expansion. Because, each material haves 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 a 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 us go and try to understand the physics step by step.

(Refer Slide Time: 02:58)




The first physics is the electrical physics. So, in the 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 are going to pull the ground to somewhere over here. The voltage with which we give could we 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 a little bit experienced you can go ahead and make a blank model.

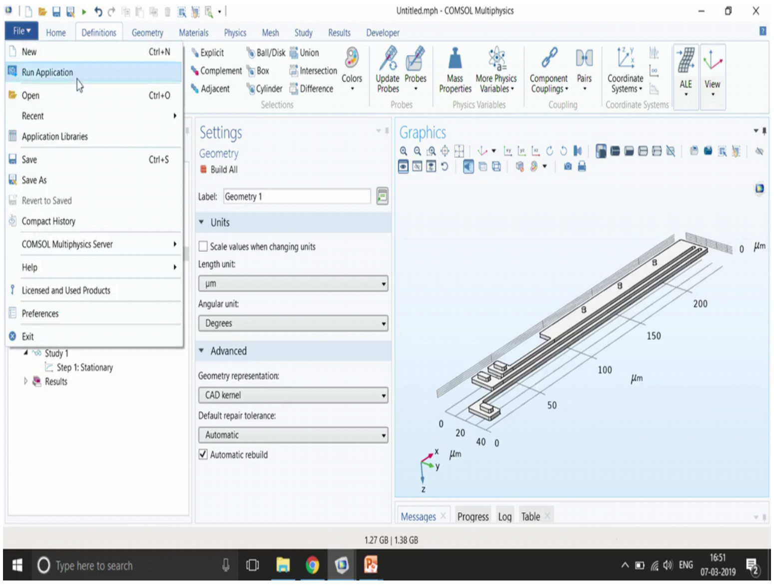
So, you do not need to add any physics just a one click will lead you to a complete new model; a blank completely model. The model wizard is a step by step process to make or initialize your model. So, let us click on model wizard and it is asking us for the space dimension. Whether you want to be make a 3 dimensional object 2 D, 2 D axis symmetry and asymmetry 1 D or 0 D. So, let us go ahead with 3 D.

Now, it is asking about which physics that you want to add. As of now, as of now we know that there going to be 3 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 mechanics physics which is going to lead to a deformation. The first thing that we are going to add is only the electrical physics, we will see how the potential is wearing throughout the your domain. How is the current getting accumulated at some part of your domain which is actually getting lead, which is actually leading to the thermal losses and that losses is eventually going to go to the thermostat.

So, our first step is to do only the electrical physics. So, let us go into the AC DC module right. So, this is the AC DC part. And in this AC DC part there are lot of different-different interfaces electrical current interface, magnetic field interface, the rotating machinery magnetic 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 conductor. If you are talking about capacitor then you need electrostatics. So, let us go with electric current 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 a steady state analysis that is only give only a particular voltage. Then we have time dependent 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 where in we give a particular dc bias along with the AC component.

(Refer Slide Time: 06:25)



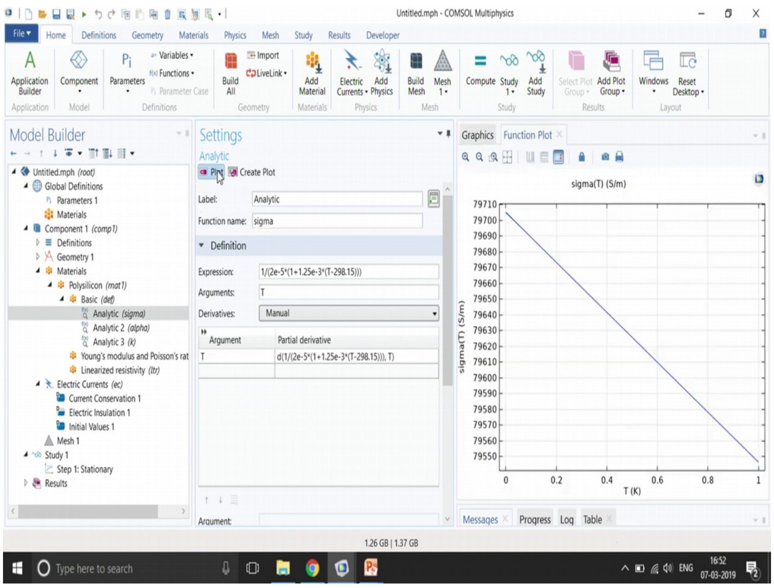

So, let us go with a stationary analysis and click on done. So, the approach that you can follow in COMSOL is the top to bottom approach. That is your first create the geometry, add the material, add the physics then perform the machine then you do which kind of analysis you want to perform and finally, the results. So, this is the top to bottom approach that usually is preferred in COMSOL. One more approach is preferred is in the ribbon plain 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 meshing, analyse which kind of analysis you want to perform, then finally the results. That is to analyse your results. You also have one more block as developer that is if you want to write your own code along with the code that is 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 cad tool and if you want to insert a sequence. A sequence is like a step by step process to make geometry.

If you want to insert that sequence from a prebuilt COMSOL file you can use insert sequence. So, let me choose insert sequence and I choose the model that I have already created. And I click on zoom extents this is the geometry that it is already available in the library file. So, if you want to go through this example model which is also known as thermal actuator you can go to file, application library and just search for thermal actuator. And you see the thermal actuator modified available 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.

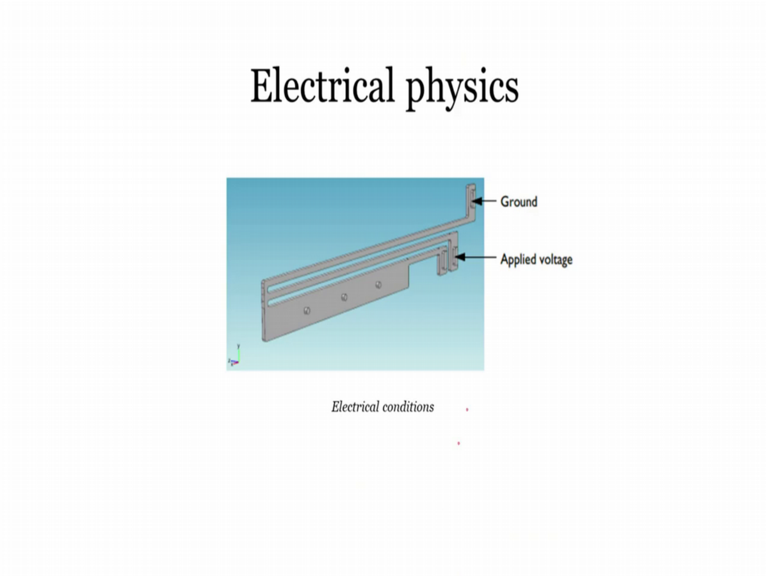
I can go ahead in the built in, I can have the poly silicon material property. Now we can see that over here the material property. It is including a electric conductivity which is being defined as sigma as a function of T. This T is the temperature. What it means is that the sigma is not constant it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

(Refer Slide Time: 09:44)




The next thing is to give the physics boundary condition.

(Refer Slide Time: 10:04)




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 and 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 V naught.

So, right now you can see that it is showing unknown variable because we have given a variable name, but we have not defined in 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 a sequence of geometry steps. So, this all step or parameters are defining the geometry steps. Let me define now V naught which is around 5 volts that is the operating voltage.

So, now the error has cease to existence 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 ok. 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 should be good enough no.

Now, you can see that your actually the dimple was cylindrical, but over here it is not even a cylinder right. So, this mesh might not be sufficient enough because your geometry might have might there in the cylinder, but the physics is going to be solved only how the mesh is being. So, it is actually is going to solve the way the mesh behaves or looks like. So, we need to go ahead and also increase the meshing let me go to finer and now you can see that it the mesh has nicely resolved the cylinder right.

So, now, this is what looks good to proceed further. So, let me go to my study one let me the right electrical study. So, my study one is electrical study where I 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 poly silicon.

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 colour current is flowing through, one of the way to do that would be right click on the results 3 D plot group. In the 3 D 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, current density and click on plot.

Still now I am not able to visualise the current properly ideally the voltage which shows that it is very high over here and it is decreasing like this right. So, current profile should also be similar to the gradient of your potential. The better way to recognise it would be to create to change the grid points. So, let me just go ahead and change the grid points hopefully we would 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 1. And now you indeed are able to see the current flow.

So, now, you can see that the potential which was applied very high over here is actually making sense. So, the current is actually flowing like this. And it going to turn over here and then it is going to move back towards the minus x direction to the ground where you have given. So, this actually makes sense the results make sense. So, it is always good that once you actually get the results you also validate the current profile how much voltage are you getting to revalidate the whatever input that you are giving is actually being consumed by your system ok.

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 3 D plot group and I can create one multi-slice plot. Multi slice would be somewhere in the more plots over here and I can search for norm of current density. So, I can search about current density and I show over here. And now you can see that the current density is very high somewhere over here. That is in the thin leg over here and over here. This means that the current at this particular the current density at these particular legs are 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.

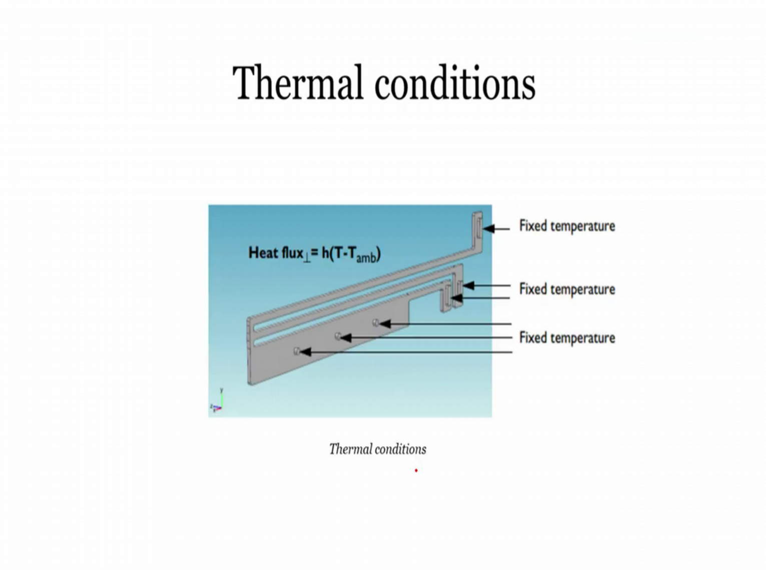
So, let us also see now if there are losses because of the joule losses how I can evaluate this. One way to say it would be again the multi slice plot. So, I can create 3 D plot group by right clicking on the results and I will create some more multi-slice plot and I search for losses. I have volumetric loss density and volume is that is complete electromagnetic and then only the electric part. There is no magnetic component so, we can choose any of any one of them. Let me choose Q h and click on plot. And indeed that is what it is showing me. The losses are very high at both of the legs you can see via a thin legs. It is even high at the corners. And that is 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 metre cube.

Now you still have the losses, but you do not know the losses are going to lead to what change in the temperature. This system is working with which ambient temperature you do not know that right 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 5 volts in room temperature. Before to 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 one is electrical analysis. 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 again I go back to my poly silicon earlier only the electric conductivity and the permittivity were having the 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 conductivity has been activated.

Again you can see that the thermal conductivity is a function of temperature. The capital T now is actually coming from here once we do the coupling. Even the conductivity the electric conductivity as a function of T, the capital 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.

(Refer Slide Time: 20:40)




For example the dimples 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 convective heat flux with the 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 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 right 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 can also going to add the convective heat flux. To do that just right click on the heat transfer in solids and you can use the heat flux boundary condition. So, I choose those and I just click on my graphics left click and then I use control a that selects all the boundaries. And now I need to deselect the boundaries of the anchors and the dimples.

The next thing are we giving a invert 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 5, which means that there is a airflow not even an air flow actually there is a air surrounding this particular system.

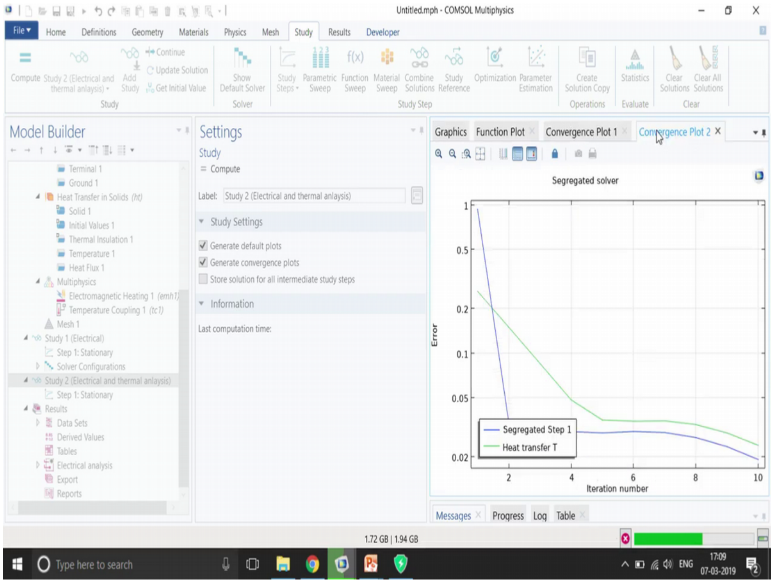
If there is a air flow the thermal expansion the heat transfer coefficient would increase. If there is water which is cooling the system it will even in cruise even faster. So, this kind of heat transfer coefficient actually depends upon what is the ambient temperature of what is the cooling condition that you are trying to give. For example, you can also give over here external natural convection, internal natural convention, external force convection and internal force convection; we saw all the option that are available. So, we have 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 it 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 electrical physics; now, coming to the coupling part. How the electrical physics and heat transfer physics are getting coupled? That if you would have been careful enough you have seen that this multi physics node was not coming up before, but right now it is coming because there were 2 physics that would be coupled. There are many automatic coupling in addition to that there are also manual coupling. Just so if you want to add the automatically coupling just right click on the multi physics node and you can see over here the electromagnetic heating.

So, let us 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 process 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 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, you can now go ahead and add one more study.

So, I am 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 both now. And you can see that both have been activated over here with the multi physics node. It is very important to note your 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 material property then the results also will also deviate. So, it is 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.

(Refer Slide Time: 26:42)




You can also see the convergence plot with the error. You can also manually tweak the amount of error that you want to use that is the related tolerance error that you want to use to make the system converge. So, now, you can see the first dot is electric potential, the second plot is the temperature and as we know, as we also have seen before that the temperature rise would be somewhere over here. Why? Because, if you see the electrical analysis in the thermal sorry in the losses plot the power losses plot.

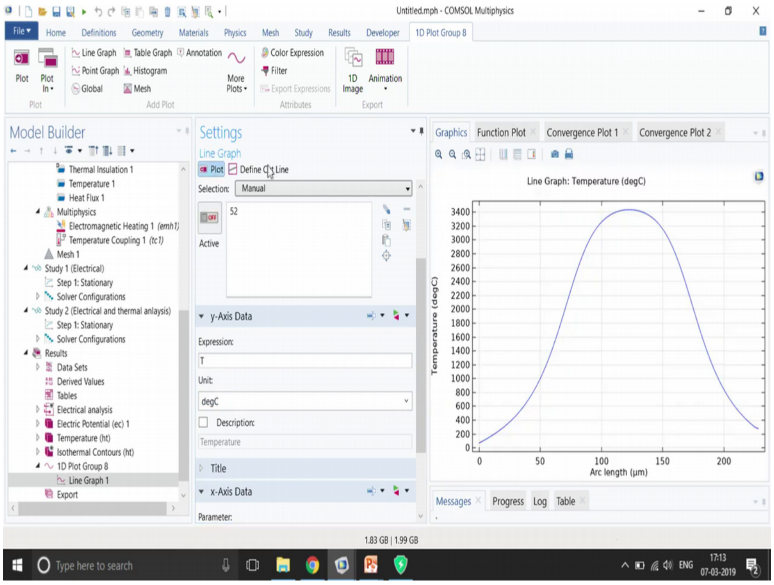
This is the power loss plot that you can see over here there are lot of heat losses over here. So, this is the temperature plot. Now for example, you want to see the maximum and minimum values over here you can see over here you want to see the actual values without the exponential part you can actually come to number format and you can use the precision values you can increase that. So, like you can see that it is very very high temperature.

And it is 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 the part that we have to do by our self. For example, if the melting point of our system is around 2000 Kelvin right. So, COMSOL will not tell you that it has it has like the temperature has closed you 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 is gone more than the melting point and this is how it is actually stops.

Now for example, now you know the temperature and you want to now see the temperature plot from this point to this point to in a line segment. How is this the spatial distribution to do that I can create a 1 D plot group. So, I right click on the results, I create 1 D plot group and over here I right click one the 1 D 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 when where is directly into the. So, let me use this line and I want to evaluate my temperature.

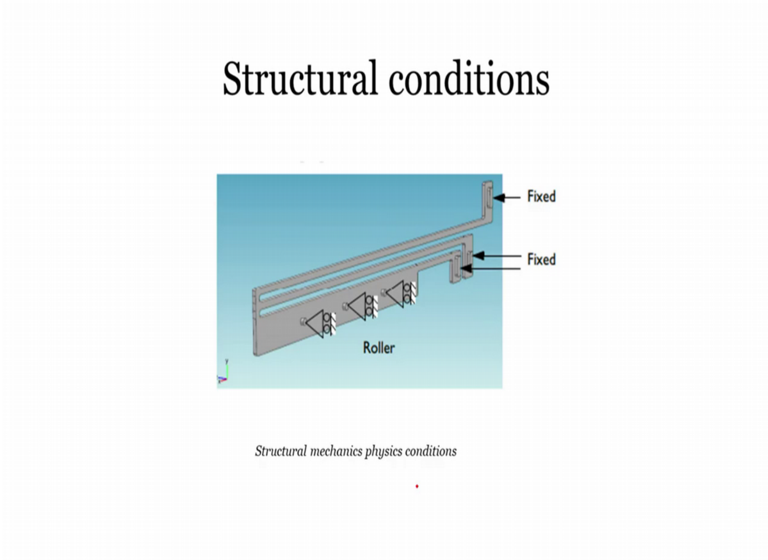
One way is to directly enter the expression over here one alternate ways to just click on replace expression and search for temperature. Before to it we have to be sure that we are using the data set of study 2. So, let me go ahead and choose my data set as study 2, that is electrical and thermal analysis and then search for the temperature. So, you can see that temperature profile variable has arrived over here. So, I 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 spatial variation of temperature along this particular line.

(Refer Slide Time: 30:04)



The next thing is to perform structure analysis. Before doing the structure 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 structure analysis. So, let me go to my presentation and see how are the structural conditions.

(Refer Slide Time: 31:05)



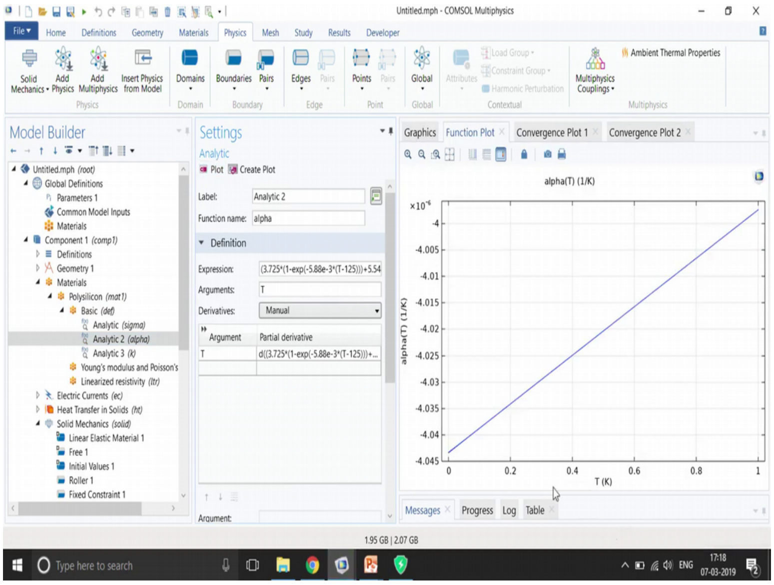

And, you can see over here there are fix boundary conditions over here, here and here that are that are on the anchors and on the dimples we are roller boundary condition over here, here and here. That means, that these boundaries can actually roll. You can you can give a particular translation motion. But you cannot give an up and down motion to the roller. However, the fix constants are not allowed to move in any direction. So, let us go and to the COMSOL and add this physics. So, we already added the electrical physics and the heat transfer physics the next thing I go to the physics add physics and I add the heat transfer in the solid. I will just double click on this and over here oh 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 the solid mechanics physics.

Again we can go to the material selection 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 is a function of temperature connectivity 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. Ok. This is just for example. Let us go with a constant value. In my solid mechanics as you have seen over here you are going to give a 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 equations. So, from the equations also you can understand how the motion is going to be. It means the normal motion of your boundary is 0, while it will allow the tangential motion. Now we go ahead with the fix constant again right click on the solid mechanics add the fixed constant part and choose this boundary. You can see the equation that u is equal to 0. 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 loses leading to the rise in temperature now we need to couple the rise in temperature to the structural mechanics using the thermal expansion. So, because of rise in temperature what is thermal expansion? The way to add that would be right click on the multi physics and then you click on the thermal expansion. The other multi physics would not be of interest to you as of now and then I add my domain. Once I add this you will see that thermal expansion coefficient is been added that is also function of T you can go within it you can see alpha over here how it is a function of T.

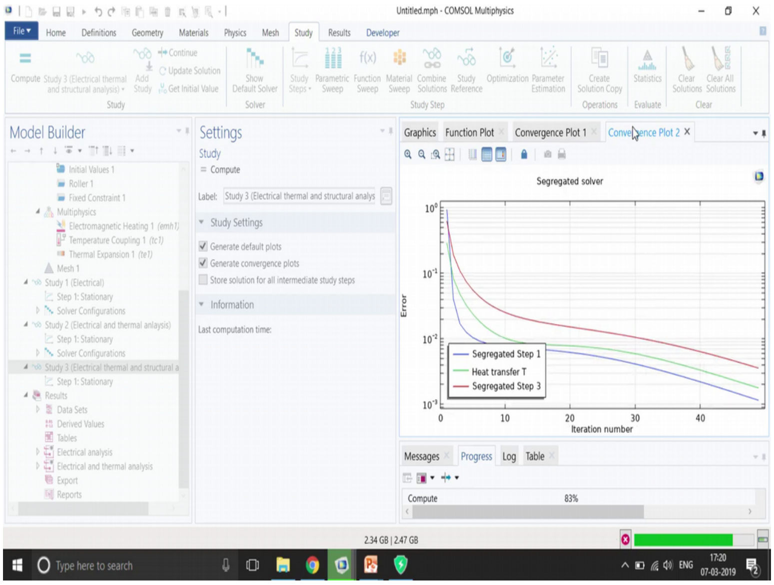
(Refer Slide Time: 35:07)




So, we have added the thermal expansion. So, we go to the study node and add one more study a stationary study. In my stationery study 3, I write electrical thermal and structural analysis. And over you can see that available all the 3 physics along with the three multi physics node. You can see over here the rest of the boundaries are free by default all the boundaries of free, but if you run your system stationary analysis that is steady state analysis with the free boundary connection your problem becomes ill post and you will get some non conversion issue which always good to assign some fix constants boundary conditions.

So, let me go to my study 3 and click on compute. In the progress bar you can see the what solvers it is using how much is iteration. So, if we are not going to cover in detail the solver configurations you can see the convergence plot over here.

(Refer Slide Time: 36:50)



If you have any questions please do not hesitate to ask your questions in the form of chat or the mails that is allowed. So, you can see the results now, you already have seen the temperature variation isothermal contours and stress plot. So, now you can see the stress which are being developed with the surface plot and if you want to see it in Newton per mm square you can just click on over here and search for Newton per mm square for mpa or you can just directly mentioned over here Newton per mm square 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 you see the displacement.

So, in that case you can just right click on the results 3 D plot group in this I create the volume and in this I search for displacement. So, you can see over here. We have to see the data set with which we are studying. So, we are doing a electrical thermal structure analysis. So, let me choose this and over here I search for displacement. So, you can see this variable solid dot disp that is total displacement. So, let me choose this and you can see that it shows around maximum. So, maximum like I can enable over here: a maximum displacement over 21.2 micrometres somewhere over here.

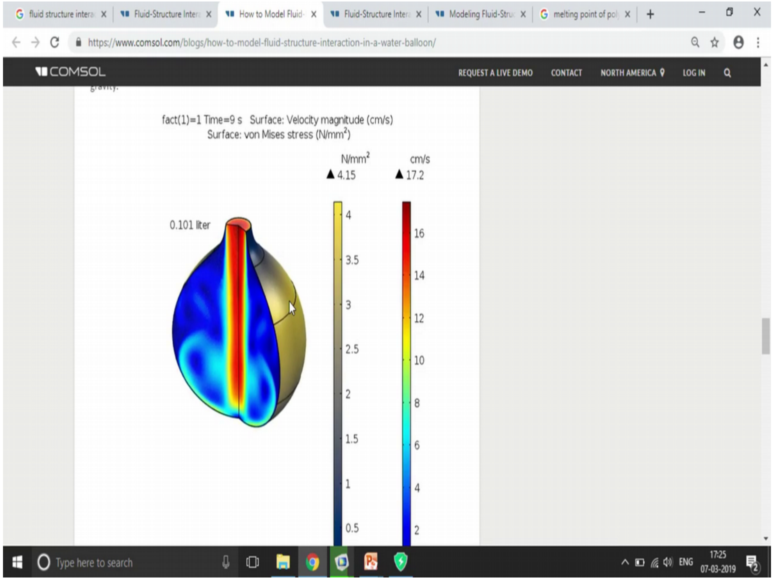
Now if you want to see or visualise how much this deformation is going to occur based upon the voltage that I have given, I can use I want to visualize how much displacement. So, I can just right click on the volume 1 and click on the formation. I can enable the scale factor as 1. So, this is how much it is 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, around 26 micrometre the length are not that big. Length sorry, let me start from this point and this point and this point yeah. So, length total distance the total length is around 200 around 200 micrometres and that particular device is bending around 21 micrometres along the along the minus y direction ok. So it is good enough.

Now; however, I want to know that if I change my potential, how much is my displacement going to very 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 poly silicon would also having some kind of a melting point. So, let me go to internet and search for melting point of poly silicon. So, it is around 1400 degree celsius. So, now I know I have some kind of threshold value with which it should not go increase the maximum temperature should 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 I can add my voltage V naught and over here I can write the range. I can write I can just click on the range I can start with the 1 volt the step of a 1 volt and a stop of 5 volts. So, it is going to move from 1 volt to 5 volts. And then I click on compute. So, till the time it computes. So, there were couple of blocks on fluid structure interaction that I will like to showcase you this is one of the block and.

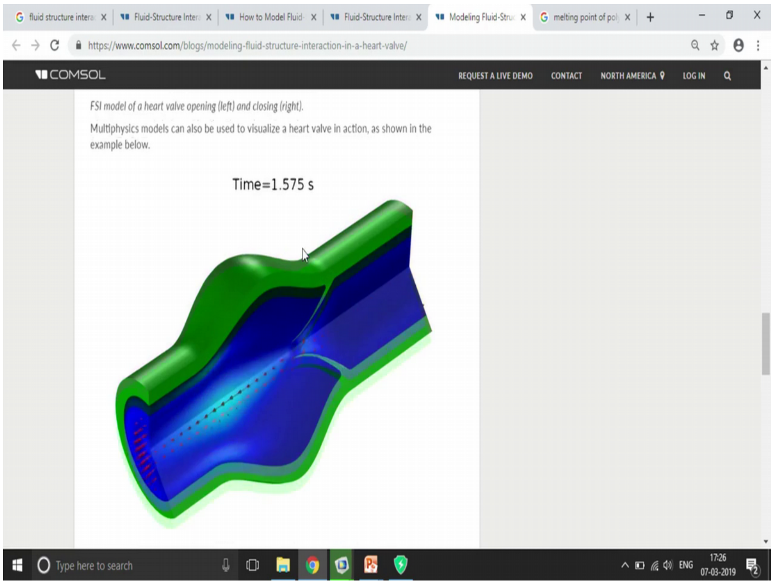
So, all the blocks are having very nice animations. So, it is it will be like very nice to actually understand the physical intuition of the physics for example, this is a water balloon during inflation what are the velocity magnitude stresses that are being developed in all those things are available.

(Refer Slide Time: 42:26)




So, this is a fluid structure interaction problem where the balloon is getting filled. Few such interaction we had one more block this is for heart valve.

(Refer Slide Time: 43:04)

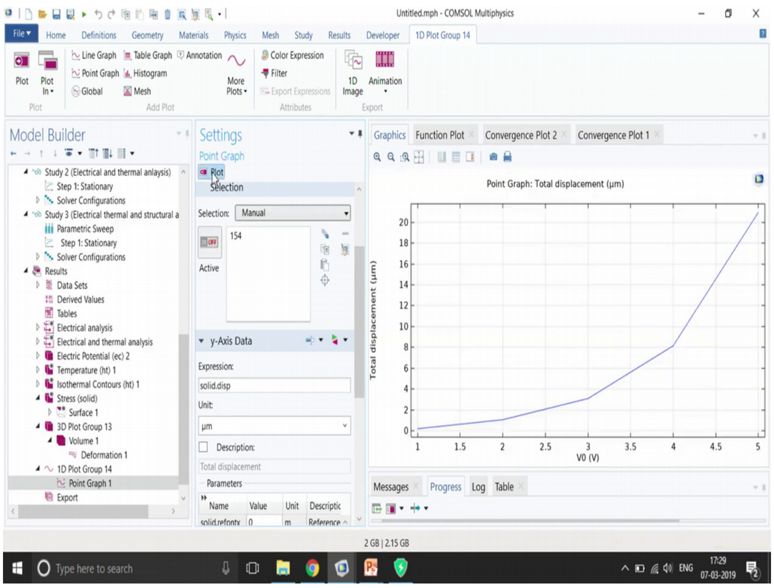



So, you can understand with these change in the flow of velocities is how the valve is actually moving. So, it requires a moving mesh technique over here. So, such kind fluid such kind of interaction modelling could also be performed. So, here you can see this what kind of solvers are we using. So, we are using segregate solver there are fully coupled solver segregated solver these are the two basic kind of solvers. Fully coupled solvers takes all the physics together segregated solver takes it step by step. There are also 2 types of solvers that is stationary sorry it is direct and indirect. direct is more robust it takes lesser time, but it takes lot of memory. The iterative solver actually takes longer time, but it actually takes less memory.

So, we have received we have got some results. So, we have now we can see that we have 5 potentials. So, in the 3 D plot group I can see for 1 volt we are getting and it will go inside this yeah for 1 volt we are getting the displacement of 0.2 for 2 volts we are getting 1 micrometres, 3 micrometres 8.23 and 2.15 like this. So, now if I want to see, if I 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 results. I can create a 1 D plot group over here, I can create a point graph and I can choose this point ok. And I can ask it to plot for once I select the data set appropriate data set that is study 3. I can ask him to plot the displacement.

So, I search for displacement I search for displacements over here and I double click on this. And I just click on plot so, it would be automatically the x axis would be parameter in voltage and y axis is going to be by displacement. So, you can see that as the voltage increasing. The displacement is also increasing and it is not it is a non-linear curve that you want to see, that you want to see.

(Refer Slide Time: 46:10)



You can also enable the markers over here. So, you can use a Asterisk marks with the in data points that is at whatever points you have swept you will get the results ok. Now the next point is to see also the temperature ok. So, that 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 sweep the domain over here and to make this max op 1 that is operator name max op 1, available in the post processing I have to right click on my study 3 and click on update solution that is it.

And over here I need to create a 1 D plot group again. So, I right click on the results I create 1 D plot group in this 1 D plot group I go ahead right click on it and create a global plot. Here I will use the same operator max op 1 is a function of temperature and I use data set as my third data set and choosing all of them. So, I will use all my x axis is again temperature. Sorry voltage. And now you can see the temperature the max temperature is increasing again as a voltage is increasing and I can add our 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 I writes 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. Ok.

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 a researcher or it could be a, you could be a student or you could be a professor who are taking this course. And you want to share this complete analysis with your colleagues.

So, if your professor you would like to give it as an assignment for your students and they want to change some kind of a parameter and run and get the different kinds of results at different operating range. Or, you would be an R and D person and you are specialized in electrical thermal and structural analysis, but you want to give it a this kind of model to your design engineers who will be designing the system.

So, they actually do not need to actually go through the complete model they are only interested in the results. So, they can actually vary the input parameters and finally get the results. They are not interested to see what is happening over here inside or you could be a student who have make this model and who is going to give a thesis presentation to a professor and they are also the professors who are attending your thesis representation they are interested to actually see different different operating voltages or operating conditions what are the different different conditions right. At that that time in prompt 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 the session how to create an app. It is very simple to create an app that is how we will show it. To do the app it is important to actually parametrize. So, I can again go to my range. So, I will it write start underscore V, step underscore V and stop underscore V. So, these 3 parameters I have defined that is my start voltage, my step voltage and my stop voltage. I will click on replace again it is saying unknown model parameter because I have not defined.

So, let me write over here as a start underscore V as 1 volt is always good to write your whatever parameter along with the unit. So, that there are no in conclusive results. Step parameter step voltage is 1 volt and then the stop voltage is 5 volt. So, it start with 1 have a step of 1 and stop of till 5 volts again it is always good to see that you have not there is no mistake in spelling mistakes and all those things.

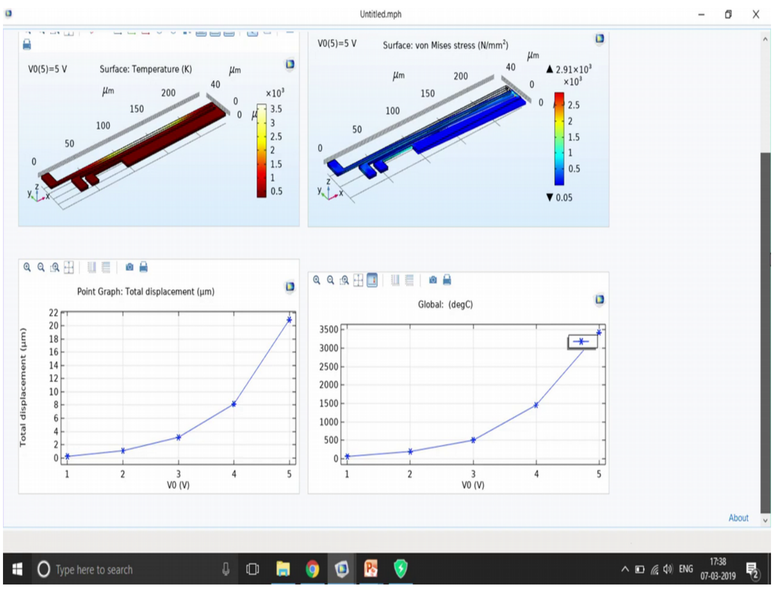
So, I have defined the parameters I know what plots I want to see. So, for example, I want to see my thermal plot over here, I want to see 1 D plot 14 and 15. So, I go to my application builder. So, only by one click I can go to my application builder. And over here I can just click on new form. Once I click on new form it is asking me all the parameters that I want to change for example, if you 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 the how the temperature is varying that also is possible.

Then the next thing is a 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 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 that is they want to perform.

They want to do only electrical and thermal or electrical and thermal and structural. So, as of now let me give you 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. So, this is my first results over here I can just move and modify all of them make it little bit large so, easy to visualize.

So, now, the compute button somewhere was over here. So, here I can just write over here and again change the text as electrical electro thermal or thermo structural study ok. So, I can keep somewhere over here, I can move my actuator length somewhere over here. So, this is my voltage conditions, this is my geometry conditions and then this is my compute button switch kind of analysis I want to do. So, I click on test application.

(Refer Slide Time: 55:12)




So, if this is 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 micrometre and then just click on this compute and where the next the next set of result should be coming up. They can also change this start voltage for example, 5 and step of 1, 2 stop of 10 and then can compute the study and see the change in the results.

You can also add the parameters as material property and also 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. So, this is how we complete come simulation of a 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 to it as soon as possible.

Thank you, have a good time.