Rheology and Processing of Paints, Plastic and Elastomer based Composites
Prof. Santanu Chattopadhyay
Rubber Technology Centre
Indian Institute of Technology Kharagpur

Lecture 47: Practical demonstration on FEA

Welcome to NPTEL online certification courses on Rheology and Processing of Paint, Plastic and Elastomer based composites. So, today we are in week 8 lecture number 8.5 this is practical demonstration on FEA. I am Ramya I am a research scholar working under the supervision of Professor Chattopadhyay and I am also one of the TAs of this course. So, in this lecture what we will see is what is ANSYS and how we can use whatever we studied before in the lectures the professor explained. So, how we can implement this in the software? So, for that, we are using ANSYS software which is a licensed version, licenses provided by IIT, KGP.

So, let me open that software this is ANSYS workbench 18 we are using here. So, this is the software this is how it looks. Let us take an example program for a simple program first which will be the sudden contraction through a pipe. So, for that let me choose fluid flow fluent.

So, here we have 6 sub-divisions these are the 6 steps which will be involved in each program. So, here first is the geometry. Geometry is the place where we will draw the structure the basic geometry what we need to solve the problem. Next one is meshing and then setup. In setup, we will be giving the input such as boundary conditions, material properties and things.

Next will be the solution and finally, will be the post processing which is called the results. So, let us start with geometry now. So, this is the geometry window here we can see different planes right x y z x y z. So, I am choosing the x y plane here then next let us choose the units we are going to use I will be using centimeters. So, as we can see this green one this green arrow corresponds to y-axis blue is z and red is x.

So, we are going to see a problem on sudden contraction through a pipe for that we are going to choose a pipe with one will one pipe will have wider diameter another one will have a narrower diameter. So, for that let us create the geometry going to primitives then choosing a cylinder. So, here we can see that the cylinder is in the z direction it has formed. So, I want in the x direction say let us give the input as 50 centimeter will be the length of the cylinder I need. So, here in z I will make it as 0 then let us take the radius as say 5 centimeter.

Then we will go to add material then add frozen then generate. Now as we can see this pipe with 50 centimeters in length and 5 centimeters in diameter is generated. Now we want another pipe right. So, let us go to create primitives and another cylinder. So, this also the same forms in the z direction, but we want it in the x direction.

So, let me give this also as say 50 centimeters in length. So, what happens is as you can see these forms in the same direction as the previous cylinder, but we want it in the opposite direction because this pipe should have a bigger diameter for that we will give this as minus 50. So, this is formed here. So, we will make z component as 0 and the radius we can give as say 10 centimeter yes add frozen generate. So, now we got the pipe one has the bigger diameter another one has the smaller diameter.

So, next what we can do is now as we can see this pipe is generated as two different parts you can see here. So, I want to make it as single part. So, that it will be a continuous pipe. So, I am going to create and I am going to Boolean operation. So, in Boolean operation if I select both the pipes and if I give apply.

So, now this has become a single solid body. Now, the pipe is a continuous pipe. So, another thing we want here is to. So, here the outlet the inlet and outlet as you can see it is closed. So, in a pipe this has to be opened right.

So, to open that we will go to thin surface. So, here we can select which parts we need to keep in the geometry. So, I am choosing this part faces to keep. So, that we can select the faces here. So, I am selecting this face and this face and this face.

So, we can cut out the input and the output. So, I will give as apply and generate. So, now we got a hollow pipe. So, I have given the thickness as 1 centimeter here as you can see the thickness of the pipe is 1 centimeter.

So, next will be. So, now this is only a solid. So, inside we need a fluid to be flowing right. So, for that we have to mention the fluid. So, we will go to tools and select the fill option. So, we need to select the faces where we are going to fill the fluid with.

So, I am selecting basically the inner surface of the pipe. So, we can select this one, this one and this one. As you can see I have selected the inner surface of the pipe here. So, we can apply, generate. So, now as you can see both the parts here are named as solid and solid.

So, what we are going to do is. So, this is the pipe. So, let us rename it, rename it as pipe and this one we can rename it as fluid. So, now our geometry is done. Let us go to the workbench.

Now we can see there is a tick mark beside the geometry meaning our geometry is done. Next we can move on to the mesh, we can edit the mesh. So, here we can see the options. So, we are going to choose the mesh. So, we can directly give as generate mesh.

So, now for our geometry the mesh is created. So, as you can see these triangle ones right these are the meshes and these points which are connecting these are the nodes. So, there is a option to optimize even the see now here it is showing like how many elements and how many nodes are present you can also optimize the number of elements or nodes if you want. So, I am keeping it as a default. So, here what we have to do is again we are going to select the faces right.

So, now we have to give the input as which will be the input and which will be the output in the geometry. So, this will be my input. So, I am creating a section called input inlet actually this is inlet and this one will be outlet. So, we can see the named sections here inlet this is inlet and this one is our outlet. We can also generate another named section as the pipe wall.

So, this will be the pipe wall. Now, we have 3. So, yes after giving update we can go to the workbench again here. So, our meshes also done.

Next let us go to the setup. So, setup is yeah here we can select like how many process you are going to use say I will take 4. So, basically this setup option this is where the solver is. So, here we are using the fluent solver. So, in this we can give our inputs like material and what type of flow we are going to use and such things. So, as we can see our geometry along with the meshes here.

So, I will keep it as a pressure based type and velocity as absolute then time as steady and then I will give gravity. So, here what we are going to do is since the flow of the pipe sorry the flow of our fluid will be in the x direction here in y which is opposite to the gravity right we will give as minus 9.81 right. Then let us go to models. Here as you can see there are different models here.

This is the models which are present in the fluent database alright. Then here are many models as you can see heat exchanger and a lot of things. I am going to take viscous laminar. So, that will be my model here. Then let us go to the material section.

So, here in material section we can choose what fluid we are going to take. For that there is a database in the fluent solver which is called the fluent database. So, I am going into that. So, as you can see here there are many fluids listed here ok. So, this I am going to use water here there is water liquid and water vapor.

So, I am choosing water liquid. So, as soon as I selected this you can see the properties of each liquid each fluid is being specified in the fluent database itself as you can see the thermal conductivity, viscosity, molecular weight whatever is required. Everything is here. If you want to include something which is not in the database.

So, that is also possible. So, I am taking water. So, copy and close. Close. So, now you can see our fluid will be water liquid. So, likewise the solid also we can do the same like solid here is the pipe right.

So, material as solid. So, I will go to the fluent database and select the material as solid. So, here you can see different solids are listed. So, I am choosing this as a steel pipe. So, same the properties of the material is given in the database itself copy close.

So, now we have defined our material. Next will be the boundary conditions. So, here what we can do is here I am not giving any other boundary condition just the inlet right just velocity inlet. So, as soon as I selected a velocity inlet here, here it comes like for velocity I am giving what 50 meter per second for example. So, that will be our boundary condition here.

Next will be the reference values. So, in reference values we can give as compute from as we can already defined inlet outlet and pipe wall I will give as inlet. So, that it will compute from the inlet. So, all the necessary data's reference values are already here. So, the reference zone here will be the fluid.

So, next we will go directly to initialization. I will go to standard initialization. So, here also compute from inlet. Let me go to graphics. Here we can see path lines. So, here this given as particle variables, but the input we have given here in this problem is the velocity.

So, let me select this velocity, velocity magnitude is here. And here let me select the highlighting surfaces that I want which will be inlet, outlet and pipe wall pipe. So, we can see the highlighted surfaces here. So, if I give pulse as you can see how the velocity distribution is different right. So, as soon as the diameter is reduced we can see that the velocity is maximum here.

So, we can stop and close this. Next let us go to run calculations. Here we can give the number of iterations. Say I am giving 250 iterations for example. So, let me give us calculate.

So, now this is calculating. So, the time for this calculation depends upon it really depends upon your system configuration. So, it might take quite some time. So, it will run up to 250 iterations. So, now it is done.

So, it is coming as calculation is complete ok. So, after the calculation is complete that is fine. So, let us go to the workbench again. So, here as we can see solution is also done. So, we can we have run the calculations there.

Now let us go to results. Results is nothing, but the post processing part. So, here you can see the CFD post windows open this is the post processing stage. So, here let us select the pipe wall. So, this is the pipe wall right.

So, as we can see this geometry here it is opaque. So, we want to see the streamline velocity through that. So, for that let us make it as transparent render. Now you can define the transparency level I am just giving 0.

9 and apply. You can see like this is now transparent. Now we can if we run the program we can see the water flow through this pipe. Next we will do the same for pipe wall get it 0.9. So, same I have to do this for inlet and outlet also. Apply and for outlet again go to edit and the transparency level and apply.

So, now as we can see the whole of this pipe is transparent now. So, here are different options I am going to streamline right now. So, I will create a streamline here. So, here also we should specify where to start that is from inlet.

So, after that you can give apply. So, here we can see. So, this is the inlet from the inlet the water is flowing through the outlet. Here we can see the velocity streamline velocity distribution here. So, if we want like this is the legend where we can see this is the like a scale right. So, blue is the least velocity and red here is the maximum velocity.

You can also change the view of it there are options here for that. So, next will be apply ok. Next I will go to the animation plane. This is the quick animation I am selecting the streamline here. You can also adjust the speed at which you want to see your simulation running here.

Let me play it here. So, as you can see the water from the inlet is passing through. So, when the orifice is becoming narrower we can see that the velocity concentration happens here and here when water leaves the pipe the velocity is maximum here. So, stop. So, this is how you can run a simple problem of sudden contraction through a pipe. You can save this and use it in your presentations on your papers anywhere actually.

So, next let me quickly go to the presentation again. So, basically what we saw is the problem of sudden contraction through a pipe. So, let me explain what is happening here. So, let us take this black box. This black box is nothing, but the solver.

If in the previous problem we use the solver as the fluent solver. So, this is the solver and next is the user inputs. So, basically what happens here is we give in the user inputs such as a geometry, mesh, boundary conditions, material properties those are the inputs. And we run it here the solver this that is the black box the solver will run it. And next we will get the output.

The output will be colored pictures and results. As we saw in the last problem the animation with different velocity profile we have generated. So, so, basically this is input something happens here and then we get the output. But how do we know that the output we are getting is correct or not. So, how to determine that for that we need to know what is happening inside the solver.

Actually for that let us take consider as some physical problem. So, any problem we given to the solver that is a physical problem. For example, a flow through a pipe is laminar flow, turbulent flow any flow. So, this physical problem like for example, in a polymer melt we can consider it as a polymer melt flowing through a die as in the previous lectures professor has already explained about that to us. So, next comes the user inputs.

So, the user inputs will be based on this physical problem. So, the solver that is the black box cannot read the physical problem as it is. So, this is basically we these physical problems are something that we know that laminar turbulent and things, but we have to input this as a machine readable form. So, that solver can read it for that what we do is the inputs are based on the physical problem.

So, the solver needs to solve this problem. So, for that what we will do is. So, the tool uses the inputs we give to determine it as the nothing, but the mathematical model. So, basically what is happening here the physical problem is not solved directly instead the mathematical model of the physical problem is being solved. So, this conversion we can see how this happens ok. For that we will give some physical principles and assumptions. So, what are these physical principles? These physical principles are nothing, but for example, equilibrium or conservation of mass or momentum those will be that.

And the assumptions will be for example, laminar turbulent and slip and no slip boundary conditions we have already seen in the previous lectures those will be our assumptions here. So, next is the numerical solutions. So, what happens is after we given all the inputs in the form of a mathematical it will take it as a mathematical model and then a numerical solution will be obtained. So, here another important thing we have to understand that the mathematical model is not going to calculate all the variables at all the points.

Instead it will take selected variables at selected points. For that for example, in fluid mechanics problem in the last problem what we took as velocity for likewise we can take pressure velocity anything as the variable selected variables. So, we need to know in the simulation what the tool is calculating directly and at what points. So, after these calculations the next step will be the post processing step. So, in this step as we saw in the last problem in this step is the where we got the animation that is the colored animation we got right. So, also it is important in knowing that what is affecting this physical problem this mathematical model this numerical solution and the post processing.

So, another way is also possible looking at the physical problem we can do some calculations by ourselves like calculations using the mathematical model or some empirical formulas. So, basically what it will give is we can know like what kind of output we can expect from the simulation here. So, it is also important to validate our simulation it is not just like we give a input and we get a output. So, for validating our simulation that it is right what we can do is we can do some experiments and have the data and compare it with the simulation results.

So, that is that will be helpful. So, we saw simple problem and a little bit on what is happening inside and things. So, basically what we had let us take another problem another simple problem that will be like clarity on how to use the software better. For example, we used a lot of models we have learnt about lot of models right in the previous lectures. So, how we can use it here for that let me take a blow molding problem.

So, in that in the previous problem we were talking about sudden contraction through a pipe. So, that was like a easier geometry. So, we have directly drew the geometry using the ANSYS itself, but in case of like some complicated structures for example, some customized dies like we saw in the previous lectures and in that cases it is a bit difficult to draw that in ANSYS itself. For that what can we do is we can use some CAD softwares where we can it is easier to draw in that another thing here is in ANSYS we cannot save the geometry when you are drawing it only you can make it as a mesh and then you can join it, but you can save it. But here when we are using for example, let us consider this bottle. So, this bottle has different structures right here we have a length then here the diameter is different right.

So, here there is a curve in this part and there are some lines here and the bottom is different. So, as we can see in a lot of soft drink bottles the geometry is different. So, how can we draw this? So, that for that I am using a software here that is the Solidworks. So, this software so, also Solidworks is also a licensed version we are using which is provided by the licenses provided by IIT Kharagpur.

So, here let me open a document. So, basically what I have done is I have already made a geometry here. So, I have made the geometry of a mold. So, this is the geometry. So, here is the mold of a bottle as you can see here there is a different definition here different and you can see the bottom there is a different definition.

So, it is much easier to use a CAD software to draw all these. So, this is the mold. So, this is the parison basically we can take it as a what a pet bottle or something. So, this material is pet and these are the two molds. So, what we can do is we can go to answers now file new. So, now we have come to the workbench here we can see like in the tool box.

So, these are the different types of tools that are available. So, for example, fluid flow blow molding poly flow. So, here whatever is in the bracket that is the name of the solver. So, here it is poly flow earlier we used in the in our programs the fluent database right like the fluent one next is also another fluid flow poly flow. So, different things are available here know how we have taken fluid flow blow molding poly flow ok. For that since we have our geometry already drawn in the solid works we can directly import it using this import geometry option we can import it into the answers workbench.

So, here it is. So, let us open this geometry and see. So, here we can see our mold and parison the whole assembly is here now ok. This is the this is the parison and this these are the two molds. So, now let us go back here go back to mesh use it as edit mesh. So, here we got our geometry the mesh window.

So, here so, here what we have to do is the geometry right here the parison. So, in last problem we saw we used water as the fluid. So, here the fluid will be the parison. So, the parison is the basically the polymer. So, we can take it as the pet material ok.

So, what happens is when the mold closes the parison will take the shape of the mold. So, here we have to mention we have to define this as fluid the parison will be the fluid in this case all right. So, let us go here let us select the symmetry insert symmetry region. So, we have selected the symmetry region here as we can see. And then let us generate the mesh generate mesh. So, here also the same we can change the number of mesh and things, but I am not changing anything right now as you can see now our geometry is meshed.

So, let us go to the workbench window again we have to update ok. The workbench window as we can see the mesh is also done then let us go to setup. So, here as we can see here the solver is the poly flow. So, this poly flow solver is now open. So, this is our geometry right what we had before.

So, here we can give the like the boundary conditions those inputs can be given here. So, let us now go and create a new task. So, FEM time dependent is selected. So, let us accept then we will define the molds.

So, we will create a new mold adiabatic mold, mold 1 is ok. So, domain of the mold. So, here what we have to do basically is. So, now we are keeping the right mold and removing the parison and the left mold. So, we have to do this for each mold and the parison. So, basically we are defining the mold here.

Contact condition, contacts conditions are nothing, but the boundary conditions here. So, you can select this and modify here no contact is already defaultly selected, but I am selecting it as contact. Next is the mold motion.

So, motion fixed. So, here I am going to give translation velocity imposed. So, that is 1 ok. So, so this A wall. So, if we select that in that from the taskbar. So, what it is basically is here we can input the time dependent values. So, here we will select the ramp function. So, here we have the values of A B C D.

So, basically the ramp function what it denotes is this indicates the time of the mold movement. So, this value modifying the A value let me give it as 0.1. So, this is basically what is in 0.2 the right mold will reach the center of the parison.

So, likewise ok, likewise the value of B value of B is 1 ok, then value of C let me give 0.100. These are basically the we are taking the time as a function here. So, and what time the mold will come closer to the parison and take up the shape of that. So, this is step by step process A B C D and finally, D is we are giving 0 this is because this is where the mold comes to rest. So, we will go to the So, again we have to define the mold create a new mold no adiabatic mold.

So, this will be mold 2 you can name it as left and right also domain of. So, here what we will do is we will keep the left mold and remove the parison and the right mold. So, next the contact condition those are the boundary condition. So, I want to modify it will come into contact. So, this is the contact I will select it. Next is mold motion the same what we have given for the right mold we are going to do the same for the left mold translation velocity imposed ok, modified translation velocity.

It is 0.5 this is also ramp function the same then the value of A B C D same we have to give that here it will be 0.1, next B will be 1, C will be then again D will be the rest position. So, the value will be 0 here translation velocity of y will be 0 ok, again is that because we are considering only one direction right. So, the other two direction it will be 0.

So, you will disable this option So, let us define the parison here. So, let us create a sub task. So, here I will be taking as the shell model generalize Newtonian isothermal. So, it is a task 1. So, here we will keep the only the parison here and we will remove the right and the left mold.

Next will be the flow boundary conditions. So, I will give inflation pressure since the parison here it will expand ok. So, this is the which I am selecting this. So, this is constant. Now, let me give the value here is minus 1 e to the power 5 Pascal's ok, this will be this will be in Pascal's. So, why we give minus sign here is the reason is that pressure will go from inside to the outside.

So, basically the pressure is inside of this parison. So, the minus value denotes like from the inside it is expanding towards the outside. So, that it can take up the shape of the mold. So, next will be define contacts here also we have to give create a new contact problem this is the boundary conditions.

So, select the contact wall. So, here we are selecting the right mold. So, let us go to the graphical window. Sizing the dots we will size up define layers create a new layer then comes the material data. Here we take shear rate dependence of viscosity. So, if we go inside here we can see the different models like the constant viscosity Bird-Carreau, power law, Bingham, Herschel-Buckley, cross law log log, the Carreau-Yasuda, modified cross law like different models what we have discussed before in this course we can see here. So, this is what I wanted to show you like how you can define your model we are giving the model only for the parison because here we are considering our parison to be the fluid.

So, that is all actually this is actually quite a big problem. So, what I wanted to basically show is. So, what I wanted to show is how this can be done using the ANSYS poly flow software. So, this is done now. So, this actually takes more time to get processed on all depending on the system configuration still there are many steps like post processing and all. So, due to the time constraint I am not I could not show this now.

So, I think you got a clarity about how to use the software and things. So, whatever I explained about the software it is already have given the source here. So, it is given in the ANSYS website itself you can always check the website and explore it more this is not only for a fluid flow and CFD. So, ANSYS can be used both for even structural simulation also like a simply supported beam anything a vibration isolators anything we can use. So, the idea was to just familiarize you with the environment of the software I think that is bit clear now. So, you can explore the software on your own by visiting the ANSYS website. So, thank you.