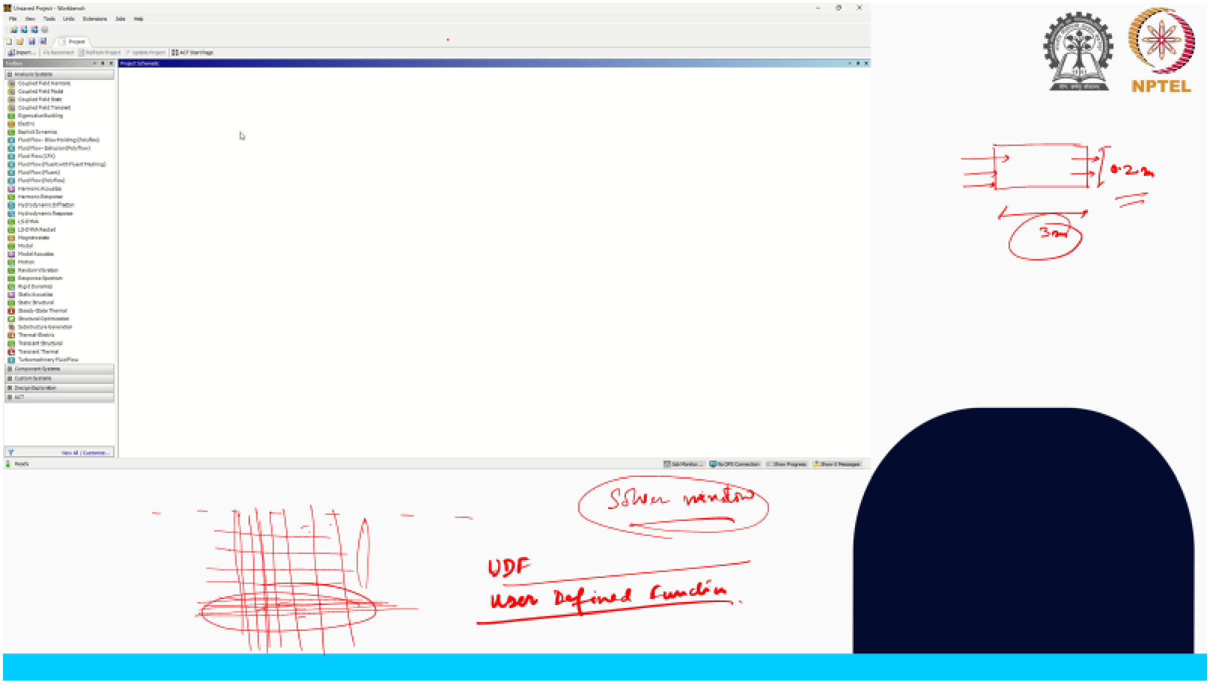
CFD APPLICATIONS IN CHEMICAL PROCESSES
Prof. Arnab Atta
Department of Chemical Engineering
Indian Institute of Technology Kharagpur
Week-07
Lecture 32: CFD Tools & Modeling Single-phase System

Hello everyone, welcome back once again to another lecture regarding CFD tools and simulating a single-phase flow problem in CFD applications in the chemical process course. So, what we are doing currently—in the last lecture, we also started simulating a single-phase system, taking an example of a 2D laminar flow problem in a pipe where we are trying to simulate or we are trying to looking at a different velocity contours, pressure contours or the velocity magnitude etcetera. The utility of the CFD models or tools has already been discussed. We also now have sufficient background to understand these operations that we are implementing by simply clicking the menu buttons—different buttons available in commercial software.

Here, we have taken an example of Ansys Fluent—how to implement such a 2D flow problem in Ansys Fluent—and we are trying to look at its implementation strategy. By doing so, we are also learning how a simple single-phase flow problem can be simulated. Now, the point is, by no means is this an exhaustive tutorial. You can find, in fact, in the ANSYS tutorial guide or ANSYS videos available on YouTube and other places, a detailed look at all the operations or your choice of areas on which you are looking for further exploration. Those areas you can explore on your own. This is merely to give you an overview of what a commercial software like Ansys Fluent looks like.

And if we try to simulate a simple problem, what options are available, and how we can do this. But most importantly, what we are trying to understand is how the theoretical knowledge we discussed before this section—learned from the fundamentals of the finite volume method—helps us, as essentially ANSYS Fluent is based on the finite volume method. So, how does that theoretical understanding help us in choosing appropriate menus or options in these cases? So, it’s not just about using any given software as a black-box option. So, in this regard, if you remember, we started by looking at or selecting ANSYS Fluent as the work option. From there, we went to the geometry, and in the geometry, we entered the Design Modeler.

Case and from there, in the Design Modeler, we eventually drew the geometry and not only the skeleton, which is the 2D domain that we had started with. We had a problem statement where the D was 0.2 meters and the length was 3 meters. So, for this problem, we are looking at a fluid flowing through a 2D pipe. Here, we are trying to simulate this scenario. So, here We drew the geometry in the last lecture. We saw how to draw the geometry and how to change the domain dimension once it was done.



But the point became, after this, we went back again. So after this view adjustment, we went back to the workbench because now we are set with the geometry part. So in the last lecture, we stopped at this point, noting that there is a tick mark beside the geometry option, which means our geometry has been created. Now, the next step is to draw the meshes because the calculation has to be done. After meshing, we will proceed to the setup, which means setting up the problem with the given boundary conditions or suitable boundary conditions, and then we will solve the problem.

Now, the point is that in this meshing, There can be user-controlled mesh options that we will now see, which are available here. So we see this stick button is there. Now, the option is the mesh. We will then click on edit, and another window will open where you see the process starting. The meshing option here shows that there is a starting meshing option, and you would realize that So here, you see there is a starting meshing option, or the working process is going on, starting meshing, and then this window will open where again you will find the geometry. So here is the geometry with the description or the dimensions.

Now here, there is an option by which you can generate a default mesh, which is just done here. So under this mesh, once you open it, you will find this mesh option. So once it is done, then if you click generate, which is done here. So again, let's have a look at it. So if I say, if I see that this mesh and then again this generate option. Once we do this, Fluent automatically makes these grids or the cells options, but this is the default option.

Now, in these default options here, you see that there are different things that can be noticed, like the physics option. So here, what you see are different options that you see here: solver preference, element order, linear element size. Now, this is the default element size that has been given. Now, but the point is that this default option looks very coarse, and we have to refine it; we have to redo this. So, we will change this element size option, okay?

So the point is, by default, what you have is that ANSYS meshing has some default meshing, default sizes. So if we generate this, click generate on the default, then it creates a default meshing. But we have to modify this meshing to further decrease the mesh size. So we will change that default size and generate the mesh again. OK, so so for that what we can see that in this options, this mesh options, we can see that is a element size.

Now we say change this element size. So these are the default options and the default result of it. But then we change this here with the default size. So, this element size is then changed to certain your desirable value and in this case say we give as say in the case it is if we generate it with this value again. So, here we are considering it as 1 into 10 to the power minus 2.

So, with this again we will click the generate and the mesh would be generated. A fine mesh would be generated because earlier that default value was quite coarser or quite larger. So, once again we can have a look at the size of the meshes now we see that structured meshes the evenly equispaced meshes that has been generated because this is a simple geometry. So, and the point is that you can rotate your mouse holding the left button or the control button. This actually is the software specific, the different functions that it enables in order to view the geometry.

Like those who have worked on the CAD geometry. Similar to that, since this is a platform where the geometry is designed and the meshes are created, you want to have a look at geometry. Every corner, every possible places in the domain vary in a minute details so that you can understand whether there is any skewness or where something has gone wrong. And for that there are different mouse options the command are there by which you can rotate, you can zoom in, have a look at it in details. So again for the default view options you go to the XY planar view and then you can look at the statistics that how many number of nodes and how many elements are there from the statistics option here.

So that would give you that how many elements you have created here and how many number of nodes has been created. Now, this is the point where again you have to remember that as you increase the number of meshes, the number of elements and the number of nodes would increase, but that also require your more computational power. So, there that is why we require the mesh independent solution because the kind of accuracy that you are looking for and your computational power. This trade off the computational power and the time by which you want to achieve the result. So, this trade off can be I mean estimated from this mesh independent study.

So, this gives your that value and here further what you see that different other details are also given ok. So, this option of sizing here this sizing again further shows that what are the options now here with these options you can essentially control or you can essentially the users can have control over the machine because here you can see the growth rate is given whether the adaptive sizing is necessary or not here the default is no. So, these things you can change or you can modify based on your requirement and you can put the constraints of the mesh. For example, whether it require adaptive meshing or adaptive sizing, whether it requires boundary layer growth or to capture the boundary layer you need say successive reductions from the center to the near the wall of the mesh sizes.

So, those things essentially can be controlled in this sizing options. So, again there are several other options that you see here in this window. Now the quality of the mesh is shown here. So that is what I mentioned in the last lecture also that see the quality of the mesh is essential to understand because the point is we should try to look into or try to have say or try to avoid rather non orthogonal cells or the meshes near the boundaries.

Which means the boundary line the boundaries and the grid lines the angle between this should be close to 90 degree which is the orthogonal grid sections. And otherwise it creates the or not otherwise in the sense that the poor quality meshes are actually quantified by the skewness factor. And that skewness the range varies from 0 to 1, so higher the value nearer to the 1 it is not as such desirable. So, this is the point where you can check the quality of the meshes. Again this definition of the mesh quality quantification varies from software to software, you look into the guide of those or help files.

So, that which value is desired here the default target skewness is 0.9 and it must not be greater than that otherwise the quality would not be accurate or not be acceptable. And then again here the option for the inflation as I told you that sometimes you require to capture a very thin film near the wall or say the boundary layers you require very fine meshes. Now, if you continue doing fine meshes throughout the domain that would cause that would result in a huge number of meshes elements and as the number of elements increases your computational power demand also increases. So, in order to reduce that what happens if you have a scenario So if you have a scenario say you need to capture the wall near the wall you can have very fine meshing near the wall and that goes coarser to the center line.

If this is the center line so basically when the meshes are done. what happens that near the wall region you can have a very fine meshing and near the core region you have a normal size meshes that actually reduces the total number of elements in the system and also helps us to have the computational efficiency or the computational power demand Not that great if we had this kind of uniform size machine throughout the domain like the finer machine throughout the domain. So this combination of the coarse and finer machine wherever needed that can be adjusted with this inflation kind of a function and that is controlled here by this options that are here. ok.

So, how to do such kind of wall refinement for that you can refer in detail to this inflation targeted inflations or the sizing of the machines. And so, if we now look at this So, here we see that once it is done now we have to select the boundary to name it. So, that we can easily identify after the simulations are done on which plane or on which surface we are plotting some data some x y data. So, basically say here in this case

we have a domain, now we have not leveled it that which one is the inlet, which one is the outlet or which one is the surface. So, how during the simulation or the solution this would be understood by the code that is the inbuilt code of this ANSYS fluent that how it understood that say for example, the flow is happening from left to right. So, this is the inlet part that we are trying to define. So, this is the naming part for our convention we will do and then in the setup we will set those boundary conditions so that accordingly the governing equations would be solved in that manner. So, for the convenience of the user we here at first have to level the different surfaces or planes in order to identify it later during the post processing or during the analysis.

So for this we need to choose, we need to go to the selection tool and here in this selection tool there are several things are there, the faces, the vertex, the edge, surface, body, etc. All these things are here as a clickable option. So, we select the edge selection option here. So, here for example, if you go to the if you go nearer this menu it would show you that what are the options and which one is for which one. So, this is the face body node etcetera all the options are there and this is the vertex option there.

So, we go to the edge and select the edges. So, for example, we select this inlet edge. And then we have to create a name for it. So, create named selection for this. Once we do this, we then write this as the inlet and we apply to the selected geometry.

So, this is how once we level it we see it has a several color code combination this is the software specific again, but this green line here apparently if you look at it very closely here once you apply that button or once you hit that apply button it shows a green line here that indicates that this is my inlet here. Similarly, we then go for the other that is the outlet one. So, again we select that as and we click that create named selection and here we type as outlet. Now, this is again the user specific name. I have given it here outlet inlet you can give it input output or whatever the name in order to identify that this is the outlet and this is the inlet.

Again once you apply you see that this is also it shown as once the selection happens it shows the green line and then you select the other two surface other two edges also and apply that as the wall. So, all the 4 edges are now we have named and that we can look at it in this named selection. Once we click this named selection this appearance is like this and then if we select one of those say inlet outlet wall say for example, inlet it then quickly highlights that which one you chose as the inlet. Similarly, once you click on the outlet hyperlink, it shows that this is the edge you applied as the outlet and the other two you applied as the valve. So, this is for the user's convenience.

So, again, once we click that this option is then closed, we click on the mesh and we see that this is the desired level of mesh. Now, we go back to this workbench, and here we see that the tick option has not appeared yet. And that would appear once we update this option because it has not come here by default, like we had for the mesh geometry. So, now we have to update this option again. So, we update it here, and once we update, you see the mesh is now showing as a green tick option.

So, now the mesh is also done for this problem. So, after naming this inlet, outlet, and here again, one thing you have to make very sure is that in this previous step, when you name this in this name selection or the named option, that this inlet, outlet, and wall, you make sure that you do not define twice or repeat this inlet in two different places. Then it can create a problem during this update, that the meshing has been successfully done, but it would show that the mesh has not been successfully done. So, in order to avoid errors or issues here, make sure these names are unique wherever you are giving this as an inlet. So, if there are multiple inlets, then it is better you define this as inlet 1.

If there is another inlet, there is inlet 2. Like this, it has to be different names, the unique names that have to be given here. And then we come out of this, we go to the mesh box again, we go to the workbench, and in the workbench, we see that we saw last time again that the mesh was not updated. So, we update this once we update, we see that the mesh work is done for us. So, meshing is done.

So, we have drawn the geometry, we have drawn the meshes, okay, and then we follow the setup. The next set of actions is the setup, the setting of the problem, okay. So, this once we click this again, I would repeat this step that once we Go to setup, right-click it, and edit. Select the edit option. It would now open Fluent launcher.

Now here, just spare a minute. In Fluent launcher, you will have several options. This is the Fluent launcher window that you see. Here, you see that there is a 2D option that we started with the problem, then there are different options available which you can choose. So, the options that are available: double precision or the single precision solver. By default, there is a single precision solver.

So, that is why we can have here is the double precision solver in terms of accuracy. And there are some other default options like do not show this panel again display the meshes after reading and here this option gives us the flexibility of applying the simulations to be applied in parallel solver or parallel processing unit. Which means when there is a complex problem generating a huge number of elements, the number of nodes is higher in those cases. We can solve it in parallel processors together simultaneously, and then again, we can have the results in a quicker manner. So here, either we can solve it in a single processor because nowadays every workstation has multiple processors.

So, you can exploit all the processing power of the machine or the workstations by setting up these parallel processing options there. So, in this case here, we will choose the double precision. And then again, the point is that we start it, and then the window that would appear would be your ANSYS Fluent window. So, previously, whatever was done was for the pre-processing part. Now, this is actually the processing part where it is reading this geometry that we have generated.

It is reading see here you would see in the console there are several things it is understanding that whatever you have defined the walls, outlet, inlet, the surface body, different faces, different domains that you created those it is reading and it is preparing the mesh for the display. And here, there are several options now that you have and which have to be set before we can solve this problem. So here, what you see is that we have chosen the double precision model, and now this is the solver window. We solve—this is the solver window in this case. so we see here this is the solver window okay and we have to put all the boundary conditions and the strategy say for example is it a steady state simulation is it a transient simulation whether gravity is working if gravity is working so this is the information that you see the general information that comes from here the general button default options are shown here Those things are there, and in the setup, we have to understand which option we should choose. This knowledge of setting the formulation, the solver type, etc., comes from our background.

The foundation that we have studied—the fundamentals of the finite volume method and associated theory. So, I will continue discussing this solver part on the setup part—rather, to be very specific—because, as per the Workbench option, this is the setup part. So, in the setup, we have to set up the solver and the boundary conditions. We will continue discussing this in our next lecture, but until then, get acquainted with this format if you would work on this particular platform. Otherwise, the point is that this is the generic view of how these software implementations apply the strategies we discussed during the finite volume discussion.

So, on this note, I will stop here and thank you for your attention.