Modeling of Tundish Steelmaking Process in Continuous Casting

Prof. Pradeep K. Jha
Department of Mechanical and Industrial Engineering
Indian Institute of Technology, Roorkee

Lecture – 31
Elements of Mathematical Modeling in Tundish Steelmaking

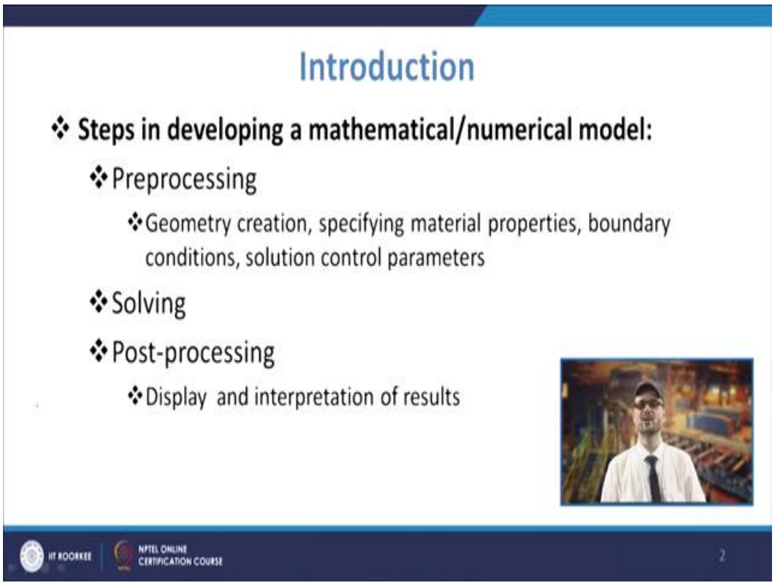
Welcome to the lecture on Elements of Mathematical Modeling in Tundish Steelmaking. So, far we talked about the prerequisites. Especially, we needed to know the concept of fluid flow, heat transfer. Then in the last week, we also talked about the solution methodology of you know equations and the solution parameters, control parameters specially, especially towards the solving of the equations.

Now, we will talk about the aspects of modeling or you know when we go for mathematical or numerical modeling of the Tundish steelmaking. So, as we know that when we talk about the models, then we deal with either the physical model or the mathematical or numerical model. Now, in those cases in the case of physical model, we need to prepare a physical model which is normally made of you know, perspex sheet or so, and then we do you know physical modeling by allowing the water normally the fluid to flow.

Whereas in mathematical modeling, you know you know we have the set of equations which are representing the case of the phenomena which is occurring inside the Tundish. And then these equations are needed to be solved and we also have studied that how you know the, the equations are solved and how you get results. Now, the thing is that being the era of computers, we normally do the computation on computers. And also for having a better understanding you know visually we have now a days tools where you can have the geometric creation, you can see the geometry, then you have to apply the conditions you know.

So, you have to take first of all these you know conditions or you have to have the assumptions and you have to also take all the conditions which will say that you are solving which type of problem. Then you are making the geometry and then putting the conditions different conditions at different places. And then you are solving the equations in that particular domain and then you are getting the results and interpreting them.

(Refer Slide Time: 03:29)



So, if you talk about the you know elements of the model mathematical or numerical model which you develop towards the you know flow in a Tundish flow and heat transfer in a Tundish so, basically they are done under the you know three head, one is the pre processing, another is solving and then you have post processing.

So, mostly the CFD tools when we use the CFD, you know do the CFD analysis. So, basically our work is divided into these three main, you know domains; so, pre processing solving and post processing. Now what we do in pre processing the in the pre processing we do the geometric creation. So, normally what we do is we have to create a particular kind of geometry. So, that geometry will be available to you. If suppose you are making the model of the Tundish so, you have to make a Tundish using a using a tool which so, by which you can create the geometry and that we will discuss that how you know you create the geometry, there are many approaches by which you create the geometry.

So, that geometry you know has to be there with you. In normal case, if you start the simple you know analysis, you can have a geometry of rectangular shape of box, if it is of three dimensional or you can have a two dimensional geometry also. So, that geometry needs to be you know clear in your mind. Then what you do is you specify the material properties, you know material of the you know Tundish by which it is made or material which is going to flow inside the you know, Tundish.

So, you will have the boundary material, material of the walls or you know the material of the the steel which is flowing inside. Then you are going to impose the boundary conditions. So, boundary conditions will be there, those conditions which are specified on the boundaries.

So, boundaries are especially the walls or the inlet or the outlet you may have blocks inside. So, and you will have a boundary conditions may be of different type, you may have the boundary condition related to flow, it may be related to heat transfer, it may be related to pressure. So, there may be different kind of boundary conditions which you need to specify in the domain. And then you are going to have the solution control parameters.

So, once you put these boundary conditions, you specify different zones as the walls or you know the symmetry or the you know periodic boundary conditions or so, or you specify the inlet and outlet and other all other things. Then after that you have to go for the solution control parameters, you have to provide those parameters which will be talking about the solution which will be done. So, basically there you need to provide you know the number of iterations, you want to go further. Then you have to you know specify the time step size, if you are going for the you know the transient type of analysis.

So, you will go for the, you know time step size that will be provided. Then you will also go for the, you know parameters which will be helping you in getting the solution converged quickly and give you the meaningful kind of results. So, that will be the relaxation parameters. So, those parameters will be you know under that solution control parameters. Now you have to go for the solving and solving most of the tools may have a separate solver.

So, that will ensure that the equations which are there integrated, because of the type of problem you have taken and because of the conditions which you have given. So, ultimately, they will be resulting into the set of equations. And these equations are typically, if you are solving you will have Naiver-Stokes equations, you have the equation for energy and they need to be solved. And then they need to be solved using these solvers. So, you may have the solving option also and you will be solving these. So, that will run and the incase of the steady state you know type of the problem you will wait till the steady state is reached.

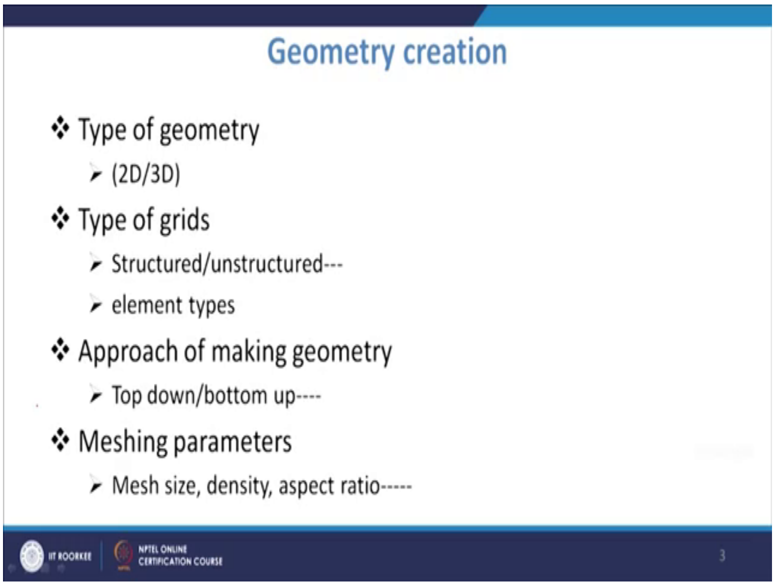
And then you have also many a times problems like the in case of transient you will see that the how you know with step size that iteration go away go forward. And then when the you are at a stage where the solution is solving process is over, then you go for the post processing operation.

So, in the case of post processing operation, you are going to have the analysis of the results. So, you have the visual display of the results, you want to have the; you know velocity or the pressure or the or computation of any parameter you know that we try to find and, and that is known as post processing.

So, post processing means after the solution is over, after the solver has done its job then you try to further see the results. And since, we have a very effective tools now a days, we can have the images, we can have the graphs being plotted for now one variable another against another variable or so. So you can have those different kind of graphs being plotted and then you can interpret the results. So, that is also you know a very good part, very important part, because whatever you get the results and then it to be interpreted in a proper manner.

So, that you get the meaningful results. So, that is why post processing also is important and there are tools available which will properly you know so, quickly it can show you the graphs of order or relationship between the different you know operating parameters. And then accordingly you can have these interpretation of the results.

(Refer Slide Time: 11:15)



So, now we will go towards the geometric creation which is the initial step you know the first step, because first of all you need to have a proper geometry of the you know the Tundish or any vessel which you are making.

So, you have to decide we are going for the two dimensional geometry or three dimensional geometry. And in that case you will have you know in the case of you know depending upon the type of geometry, you will have to choose, because if you go for two dimensional geometry which will be enough for the complete representation of the flow behavior or the behavior of the system.

In that case you prefer to go for two dimensional geometries, because that will take less amount of computational time, else you have to go for the three dimensional geometries. And then you know the geometry needs to be you know divided into small elements. And as we have studied that you need to apply these conservation equations.

So, you will have the suitable equations, you have the algebraic expressions basically you get out of you know out of these conservation equations when you apply the different kind of differencing schemes or discretization schemes. So, basically you are you know converting the whole domain into small elements.

And then you are so, basically you will have divide the domain into different control volumes. And then you are going to start the work now as far as the grids are concerned if you have to know that what kind of an analysis, you are going to have on the you know on that particular geometry.

So, whether depending upon the geometry, you have to see that whether you are for the structured type of grids or you have to go with the unstructured type of mesh. Now structured type of mesh are there where normally you know you can identify the adjacent cell with the help of the i j and k values. If you are going for the three dimensional analysis in that case, if you have to locate a particular cell that can that can have a unique i j and k value. So, that is normally the structured the kind of grid. And if you have the grid you know where it is not possible to have that kind of you know grid structure.

So, there they are known as the unstructured grid. So, in that case certainly it uses it's own internal data structures; its own internal programs. And with the help of that it will be taking you know you can go for a particular point or on a or a plane can be located, a surface can be located or a volume can be located and accordingly the analysis can be done. So, you can have these you know structured or unstructured grid then you have the element types. So, the element type you know you may have. So, you may go for 2 D structure or you may go for 3 D structure.

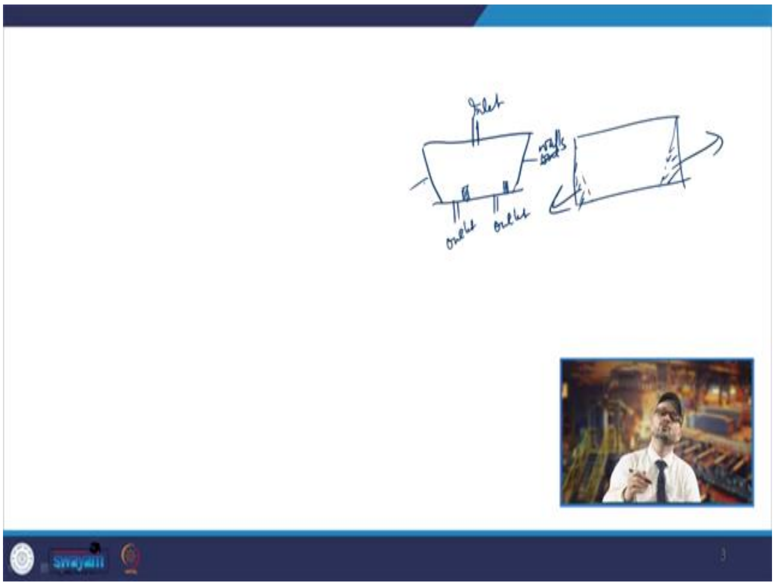
So, based on that you know if you have the 2 D type of grid so, you have you may have the element type may be like you have triangular or you may have the quadrilateral type of element whereas, if you go for the three dimensional level structures, you have different kind of you know grid topology. So, that may vary from the tetrahedron to hexahedron to you know maybe prismatic type. So, you have different kind of shapes. So, do we have may have different types of elements which you may choose when you are making the grids. Then you know approach of making the geometry. Now, here you must know that you can make the geometry in the top down or bottom up approach.

So, this is normally available with the commercial solvers. So, you have many kind of commercial you know CFD tools like phonics, fluent is there, star CCM+ is there or there are many you know CFX and you know all these. So, these are using their in their own way they are making the geometries. And you know you have solid modeling tools also like you can use the solid modeling tools solid works, catia and all that. So, you can use these tools to make the geometry. Now making the geometry is using the top down or the bottom up approach.

So, bottom up approach means, you are starting from the initial and then making the things and then moving towards the final result by adding by in the adding stepwise, addition of a stepwise like you are first of all making the points then you are making and joining the points that is by lines then you are making the faces, then you are making using the faces you make the volumes. And then you integrate them and then in that you may specify certain kind of you know zones like you may have the blocked zone where you want you want that it should be blocked or so or you can specify there itself other zones.

So, that we will see like different zones like wall inlet or so, so that is top bottom up approach. In many cases, we go for the top down approach also like, you know many a times, it becomes helpful if you have to make suppose one geometry.

(Refer Slide Time: 18:29)



Suppose, you have to make geometry where the things are suppose this way that is you know tapered. So, in that case you can make one geometry of this type and then maybe that you can take this part being off. So, that is removing this part taking away so, that will lead to this result.

So, you can start with a rectangular type of or, or a you know three dimensional structure and then cut a certain portion and make the required geometry. So, that is bottom up I mean top down approach. So, you make the larger you know geometry and then you are you know reducing that and, and getting the required geometry. So, that is using the top down approach. So, both the approaches basically are you know in practice and, and we used by the modeling tools to make a proper geometry. Then comes the meshing parameters.

So, once you make the geometry you will be making the geometry, you will be integrating them you will be making a complete you know geometry of virtually physically representing the domain. Then you have to go for the meshing of these you know, geometry.

Now, why meshing is important that we know that by meshing, we are going to have the formation of the small control volumes or elements which will be linked to each other. So, that when we apply these conservation equations and solved you know when the equations are solved simultaneously in that case you are going to have the values of you know parameters or variables in the respective cells or, or at respective nodes.

So, the meshing needs to be done properly. Now in while meshing you need to have the you know proper, you know care for the mesh size. So, you can have the large you know number of meshes. So, mesh size will be small you can have a small number of mesh so, mesh size will be larger. Now every you know as it is a quite evident that when you take large number of mesh, in that case the computational time will be higher and when you take less number of mesh the computational time will be lower, but then certainly the accuracy will be affected.

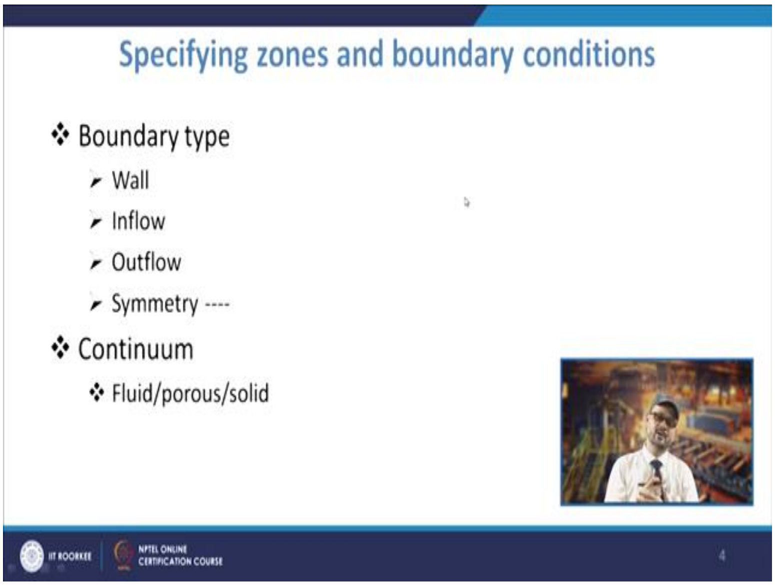
So, accordingly you have to take the mesh size, you have also to be careful while taking the mesh size, what should we by what way and the even the mesh size should vary. So, in case of the boundary layer regions, you may have to have the, you know proper variation in the size of the meshes from the walls to the point in the active domain. So, that to basically properly, you know predict the output parameters, you need to have a, an understanding of that also that how you have to maintain those you know mesh size ,then the density that is what it talks about and also the aspect ratio.

So, typically you know aspect ratio we will talk about the, you know ratio of the dimension of the you know mesh in the perpendicular directions. So, basically we try to have aspect ratio of one normally which is more ideal, but it is not always possible. So, we try not to have a very large value of aspect ratio in most of the cases.

So, these are you know points which needs to be looked into while going for the meshing. So, once you do the meshing then, it will show and you can have a look of the mesh that how the meshing has been done. So, you have to go for the net checking off also the meshes and you can in the software have the capability to ensure that there is proper mesh otherwise they may so, tell that there is no proper meshing there is no proper integration.

Maybe, once, we the situation may be such that you have done the meshing, but their the portions are not integrated. So, there will be two different zones and there may not be you know transfer of information from one region to other. So, all these things need to be checked into well when we do the final meshing at that time you need to check all those things. So, that you can go further and when you go for specifying the other parameters in the domain. So, that time you have not to face much of the problem. So, after the meshing or and after the geometry creation, we need to be looking at the specifying zones as well as the boundary conditions.

(Refer Slide Time: 24:35)



So, you know what we do is that as you know in typical you know at a Tundish you have different you know parts like now this is a Tundish so, if so, you, you will have inlet and then you may have the outlet. So, typically you have these you know these are the you know walls and then this is suppose inlet and this is the outlet. So, that is there and then you have these are, are the walls may be that many a times we use a certain kind of dams or weirs that are they are the flow modifiers.

So, what is done some of the tools you know while making geometry itself you can show them as you know you show them as the walls or the blockages.

And some of them while providing the solution control parameters or while giving the boundary conditions there you have those you know options we have to define that this is wall or so. And then accordingly you have to provide the proper boundary conditions.

So, if you look at this so, what you have to do is first of all you have to define the zone where you define the boundary type and you also define the continuum that is. So, in boundary type normally you have the, the boundaries you know specified in terms of walls, because external boundary is normally the wall. So, you will have to define those surfaces as the wall. Similarly, you will have the conditions like the inflow.

So, you will have inlet. So, that will be the inflow boundary condition will be there then you allow out flow boundary condition that will be outlet. So, basically while making geometry also, you can specify those zones as the walls or the inlet or inflow or out flow or symmetry so it will go there. And while you are giving the proper boundary conditions at that time you can have the selection from the, you know drop menu that if you take wall whatever you have specified as walls they will come and properly selecting them you can have the proper values fixed to them.

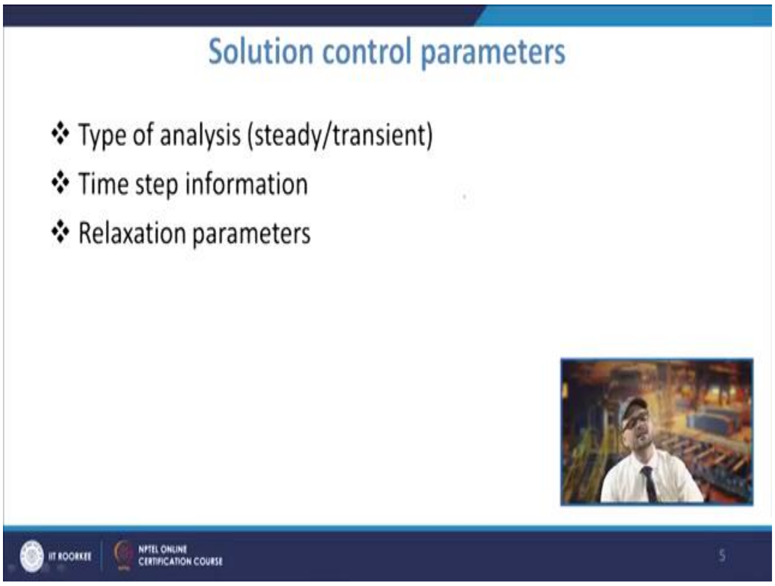
So, accordingly you know this way you provide these you know, you specify these zones as the boundary types and then you are going for the continuum or the, you know. So, in continuum you know means that you are going to define the domain wherever you have the fluid. So, you will say that it is occupy going to be occupied by the fluid. Similarly, wherever you have the, the existence of porous materials in case of porous you know flow. So, you have to define that this region needs to be specified as the porous material. And similarly, if there are blocks or solid portions so, you need to define them.

So, as I told that you can define these things either while making geometries or you can and there itself or while you are specifying the boundary conditions you can do that job you know while going for the specifying the zones and the boundary conditions.

So, once you have given those specified these zones as boundary types and also the continuum that is where there is fluid where there is solid or so, or porous media then you have to provide the solution control parameters. Now solution control parameters means now after that you are going to have the solution started.

Now, what you need for we need to specify when you have to; you want to talk about the solution to be started. So, basically, you have to tell that whether your solution is steady or transient. So, if it is steady it is fine you have to tell that for how many iterations you are going to stop, after how many traces you are going to stop and see the result.

(Refer Slide Time: 29:43)



Whereas in the case of transient analysis you will have the specification of the time step sizes. So, for in a particular time depending on the step size you will go it will go for that many you know number of iterations and you are going also to provide certain relaxation parameters. So, that parameters need to be specified you know towards getting a converged solution. So, these you know relaxation of parameters will be provided and then the solution will start and it will reach till the till you get the results. So, once the solution is over in that case after that you go and do the, you know post processing of results, you interpret the results in your own way.

So, this basically are these are the elements of a typical you know model mathematical model or numerical model which you are making especially with the help of you know you will you will be you must be having the exposure to certain of the tools modeling tools.

So, you can you have to you know proceed and you have to have the information about these things. So, that you can get the meaningful results after the post processing I mean after the processing operation that is in case of post processing. So, we will talk somewhat more about the boundary conditions and all that how it is what are the different kinds of boundary conditions which are you know imported in all that in our coming lecture

Thank you very much.