Modeling of Tundish Steelmaking Process in Continuous Casting

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

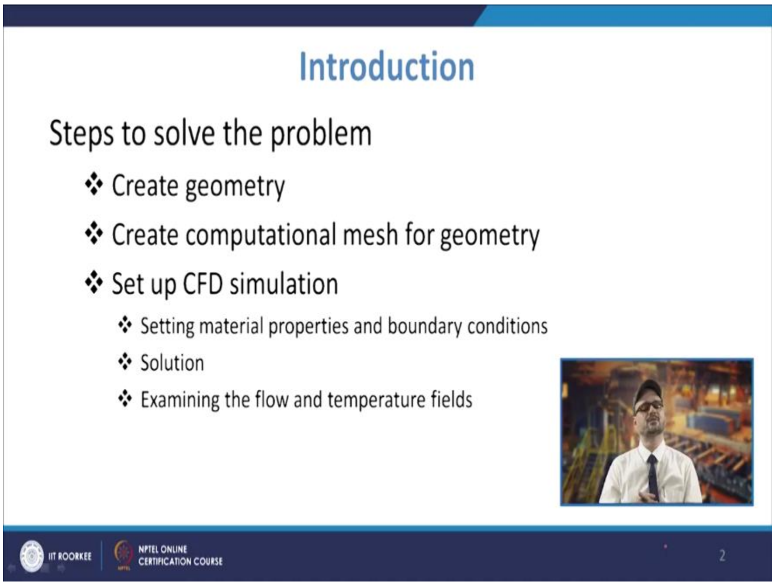
Lecture - 33
Flow Analysis in Tundish

Welcome to the lecture on Flow Analysis in Tundish. So now, we will see that how the molding work is carried out for the flow analysis in tundish and how you can show the flow behavior you know of the liquid metal inside the tundish because the inlet through the inlet the liquid material metal enters and then, it is going to all the outlets. So, once striking first of all it will strike at the tundish bottom and then, from there it will move gradually towards the outlet or outlets in the case of multi strand tundish.

So, now in that case as we had discussed that the methodology for the flow analysis is that you have to create a geometry you know of significance where you want to in which you want to have the you know flow modeling to be carried out. Then, you have to select those parameters like what is the inflow rate you know into the tundish and what are the boundary conditions.

Like if you are solving for temperature, then what are the conditions. You know at the walls what you are going to specify and what are the other parameters like turbulence parameters or may be other things, we have to set like you have to go for the solution control parameters and all that. So, and after that our job is to have the analysis of flow inside the tundish.

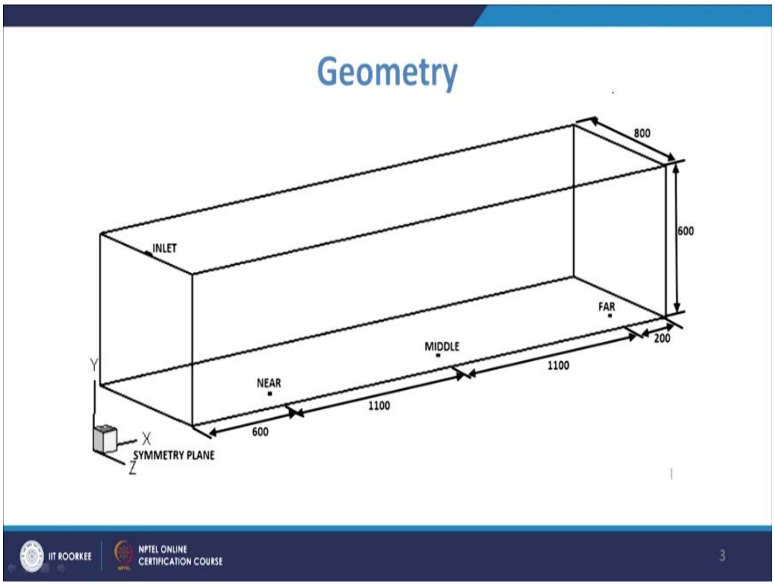
(Refer Slide Time: 02:29)



So, that is what we had earlier seen that when we solve or solving a problem, in that case we first create a geometry then we create the computational mesh for the geometry and then, set up the CFD simulation. And in that we have to set the material properties and boundary conditions. Then, we have to solve the equations using the solver and ultimately, in the post processing you know stage we have to examine the flow field as well as the temperature field and we are solving for any scalar you know we have to see the scalar transport, you know also this is the scalar field also you have to check it.

So,. So, first of all as we discussed that we can take one geometry you know hypothetically or you know you we can have.

(Refer Slide Time: 03:29)



So, on this I had done this simulation. So, in that what we see is that this is a typical tundish and as you see that this is the inlet and these are mid NEAR, MIDDLE and FAR are the outlets. So, this is typically a multi strand tundish you have more than one outlet.

So, then we call it as a multi strand outlet multi strand tundish because you have more number of outlet. We have given this nomenclature NEAR, MIDDLE and FAR because it is near to the inlet that is why; otherwise it may be taken as outlet 1, outlet 2 and outlet 3 also.

As you see that this is the length of the tundish and this is the height of the tundish that is 600 mm. All the dimensions are in mm. Suppose then, this is 600 mm is the you know height and 800 mm is the width and then if you add, so it will be 2200 plus so it will be 3 meter 3000 mm is the half length of the tundish and it is the symmetry plane.

So, it means symmetry plane means the half of the tundish is you know extending towards the left side also. So, your that way your whole tundish length becomes 6 meter. Then, the outlets are placed as you see it is near to this wall. It is not in the you know middle. So, had it been placed inside the middle, we could have even taken the one-fourth of the tundish for the calculation because that way you know that is the symmetry maintained you know.

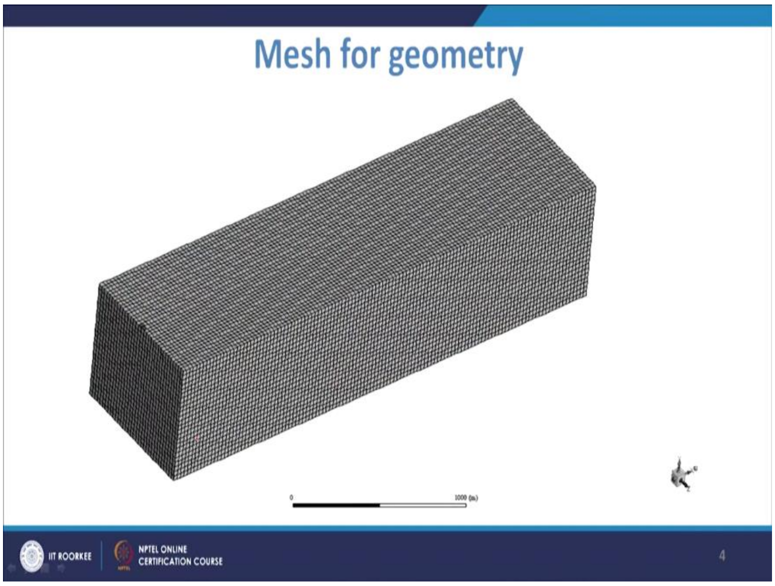
So, in that case you have to make the base geometry. So, for that in normal you know in any kind of modeling tool, you can create these geometries either by taking these points at you know different positions and then, you can you know join them to form edges and then, you can have the surface creation and then, you create the volume. Similarly, you will have to have these areas you know identified like for the inlet and for the outlets and all that.

So, that way we create these geometries in the tundish. So, after we have created the geometry, then we have to mesh it. So, you can have a structured mesh or you can have unstructured mesh, using the you know using the computational tool.

And in the structured you know mesh, you will have you will be giving certain number of you know mesh in the X direction, certain number in Y direction and certain number in Z direction. So, accordingly, you will have a structured mesh and in that as we have discussed that the trait of that mesh is that you can have any particular point or any particular cell.

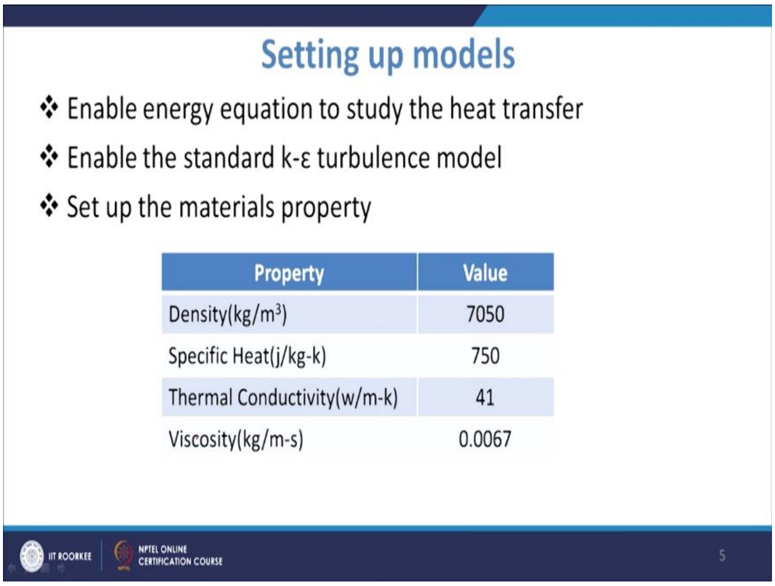
So, they are represented by a particular value of I J and K. So, or else you can have even unstructured mesh. So, far so for simple symmetries you can have a structured mesh. So, if you do the meshing and if you take these number of cells in the X, Y, Z direction.

(Refer Slide Time: 07:11)



So, accordingly, you can have this kind of you know gridded you know domain; the domain is now filled with the number of grids and it shows the you know control volumes which are formed because of the you know the number of meshes which have been taken in the three dimensional three-dimensions. So, you have a three-dimensional geometry. Now, and it has also the grids. So, that way you get this structure.

(Refer Slide Time: 07:51)



Now, you are going to set up the model and you know before that you may identify the different you know different zones also inside. So, you can have the identification of inlet, outlets. Then, you have the different walls, you can set as the walls and also you can have the you know zone also will as the continuum or where the liquid is there so that you can set it.

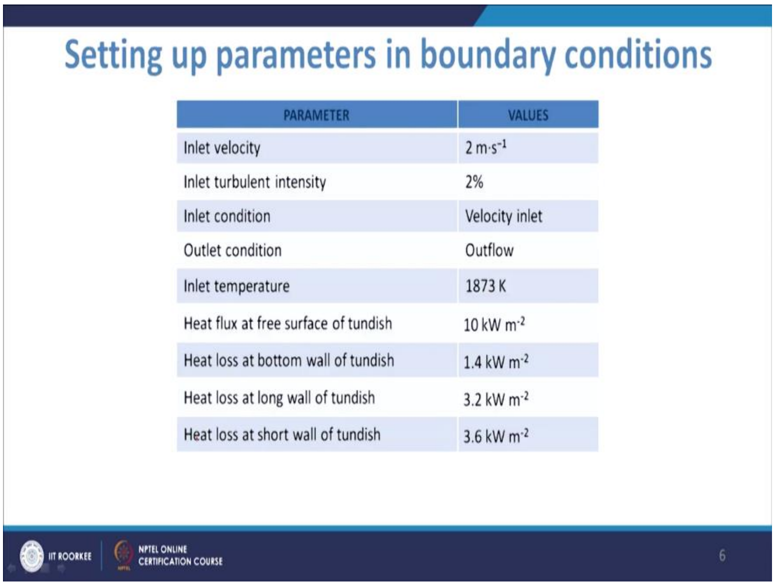
Now, what you have to do is you have to set up the model and in that if you are going for the heat transfer study also you have to go for the enabling that energy equation. So, you have to solve the energy equation or else you will write the equation of energy and you will be using the standard k epsilon turbulence model.

So, you have to you know write those equations, if you are solving with your own hand written code or if you are using any computational code which is freely available or it is available with you. Then, you know you have to select them like you can you go for a standard k epsilon turbulence model or you can go for all you know different types of turbulence models which are available, like may be R and G and realizable k epsilon and you know other low Reynolds number models.

But that depends upon the for simple you know type of flows standard k epsilon turbulence model works better. So, we go for the standard k epsilon turbulence model. Then, what we have to do is we have to set up the material properties. So, as we know that we are going for the steel. So, you can have these approximate value of the density, specific heat thermal conductivity or viscosity of the steel.

So, accordingly you that these values may be required you know for solving the equations.

(Refer Slide Time: 10:11)



Now, when you are going for the boundary conditions, so, in that boundary condition, you will be having the option to take the values at those boundaries. So, you will have the inlet condition and in the inlet as we you know if you know the you know casting rate or if you know the you know mass flow rate which is going from the ladle to the tundish in that case depending upon the you know inlet diameter or inlet dimension, you can have the calculation of the velocity at the inlet.

So, that is known as velocity inlet and in that case, you can calculate the velocity. So, if you know that mass flow rates, it will be you know ρAv and ρ, we know for the steel and a is the area of the inlet. So, accordingly you can find the velocity. Then, you know turbulence intensity as we have discussed you know we have the expression for the turbulence intensity and that is defined in terms of percentage.

So, and also it is basically representative of the you know turbulence which is normally 2 percent to 5 percent value is taken and it will be depending upon the fluctuating component you know with respect to the mean component of velocity in turbulence. So, we normally have it 2 percent you can also have the length scale. So, length scale also as you know that is normally 0.7l; characteristic length of the tundish. So, like that you can have those values.

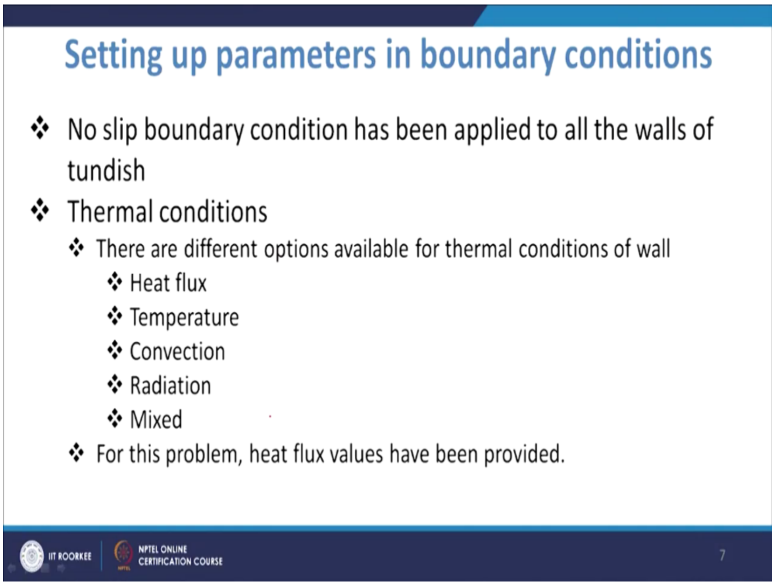
Now, as we know that we have taken the inlet condition as the velocity inlet, means we have given that velocity particular velocity at the inlet; then, you have the outlet condition as the outflow conditions and you know there is a mass balance that will be maintaining and the inlet temperature if you are solving for the you know temperature. So, in normal case you take that inlet temperature value. So, that is we set some value. So, maybe if you have a metal which is entering into the tundish of 1600 degree centigrade. So, you can take the you know inlet temperature.

Now, as you know that you have the heat flux that needs to be provided. Now, the thermal conditions include the you know can the boundary conditions at the walls because from the walls the heat is being dissipated and also from the top surface, the heat is being dissipated. So, from the top surface it will be going you know to the atmosphere and from the walls also there will be we have discussed that there will be different way by which the heat will flow through the walls and then, it will go to the surroundings.

So, for a simple case you can have the you know heat flux from the free surface at the bottom wall, at the long wall and at the short wall of the tundish. You can have those you know long wall means along the length and then short walls are on the sides. So, from there you can have these you know heat flux values that you can take and that will be required to study the temperature distribution, how the temperature changes.

Now, when we are; so, as we discussed that we are you know applying the boundary conditions and normally on the walls, we apply the no slip boundary conditions.

(Refer Slide Time: 14:29)



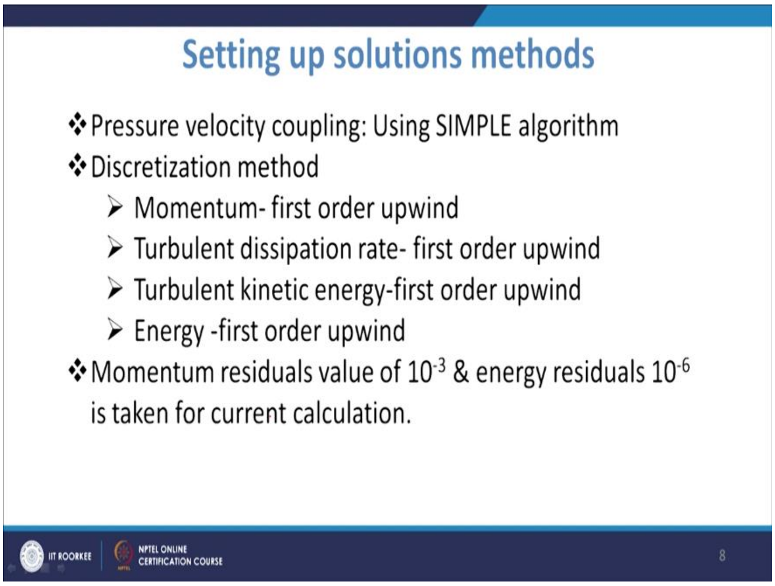
And we also use the standard you know log wall function in the case of you know standard k epsilon turbulence model.

So, in the standard k epsilon turbulence model, you have to also be you know careful that the first grid where it should be placed because we are not taking the roll low Reynolds number values you know low Reynolds number flow. So, we are taking that to be a turbulent in the turbulent region. So, you will have to have the first grid in that zone you know so that care has to be taken. So, and then in the thermal conditions, you may have the different options and you can take these heat flux condition as we have taken in this case.

Now, when we talk about the wall boundary conditions, then many a times we can take the low Reynolds number models also. Because they you know in the vicinity of the wall, they take the condition that the flow to be laminar you know and in that case you will have the more number of grids near the wall and that is why you know they sometimes take larger time. They take larger time, because we want some number of grids will be more in that case that time requirement will be more.

So,. So, that way you are these wall boundary conditions are to be provided you know with proper understanding that what kind of model you are taking and how you know that will be helpful in true manner, the flow which is you know occurring inside the domain. So,. So, that is about the wall boundary conditions.

(Refer Slide Time: 16:35)



Now, once you have set all these boundary conditions like inflow, outflow, wall boundary conditions, then you have the thermal boundary condition that may be given you know in terms of the heat flux value or maybe you can provide a specific temperature also so that will way it will calculate the heat flux value. In this case the it calculates the temperature value.

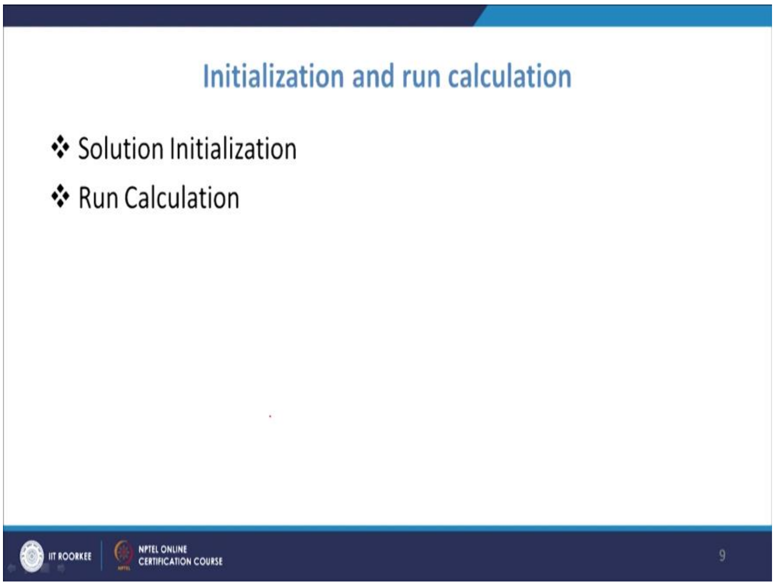
So, accordingly you know these values can be given and then, now before we start the solution. So, we have to set the solution parameters. So, before that how we are going to you know take the pressure term, you know into account. So, for that you know there are different algorithms and then, you we use the semi implicit you know based algorithm pressure linked equation. So, so that is used. So, you have you have simply simple you have simply you have p. So, different type of algorithms are there.

So, you know pressure is very important term that minus $-\frac{\partial p}{\partial x}$. So, $\frac{\partial p}{\partial y}$that term is coming. So, we need to you know model that also. So, for that you have these different algorithms and normally simple way take. So, you can have the from the drop down list, if you are using one you know code and then, also there also you have to define that how you are going to discretize take the discretization methods; maybe for the momentum, for this specific rate, for kinetic energy for energy.

So, normally we can take a first order or second order upwind you know discretization schemes for them, we had a certain idea in our earlier lectures. Then, we are also taking the residuals because you know the residuals this. So, you know when you have to stop the you know solution. So, basically that will be only when you feel that the steady state is reached, when the there is change in value which is less than certain you know set parameter, set value.

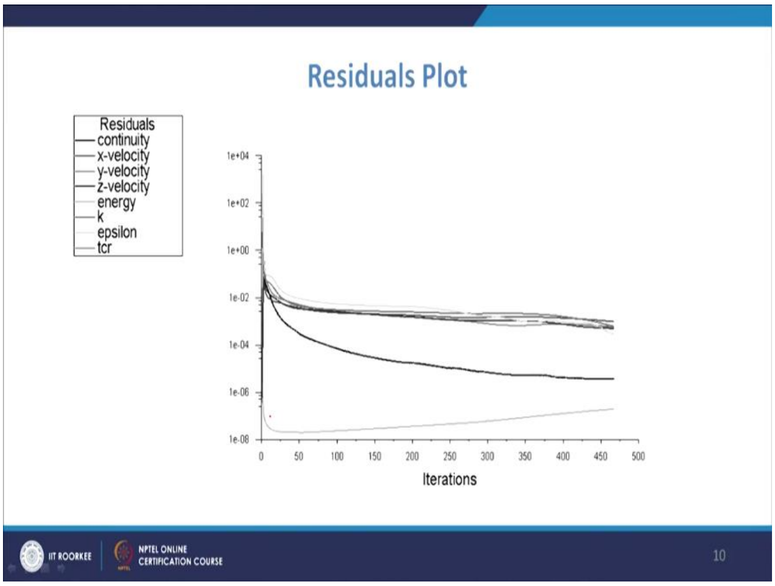
So, because you know in that case when there will be no change and you are setting that there is a change is less than certain value. Then, you are stopping. So, we are changing giving certain values in standard you know codes and then, we get set these value and you can have you know 10^{-3} to 10^{-6} for these you know parameters.

(Refer Slide Time: 19:29)



Then finally, we do the initialization for the starting of the solution and then we start. So, normally we initialize from the inlet values and you know values are initially provided and then, we do the iteration and that iteration goes on and when we are doing the unsteady analysis or steady analysis, in that case time is not involved. So, we give the data for how many iterations you have to start, we have to run the program. So, that you get the proper flow field.

(Refer Slide Time: 20:09)



So, if you run, then you get these, this way the residual plots.

So, as you see we have set the residual plots and this way it will come and after that after some time, certain time if the residual conditions are met, as you see for continuity for x-velocity, y-velocity, z-velocity generally energy turbulent kinetic energy epsilon is the rate of dissipation of turbulent kinetic energy.

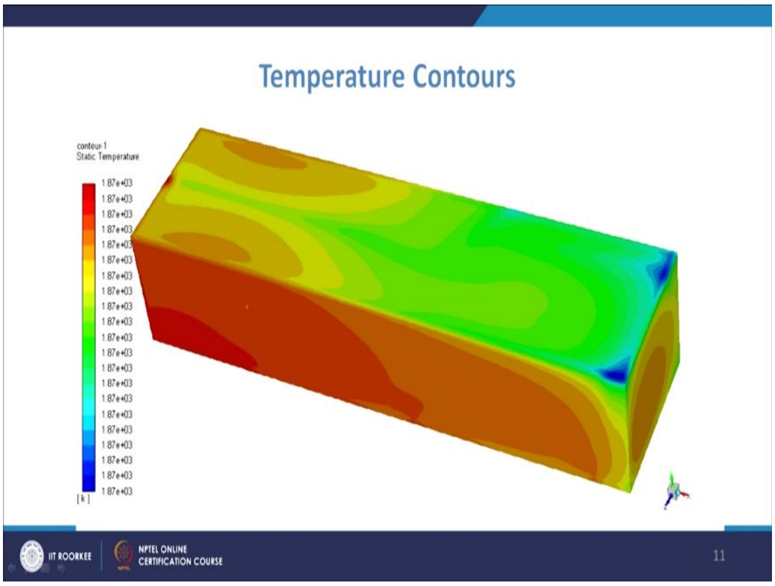
And then, you have if you are solving for even for the tracer or scalar quantity in that case that is also solved. So, that way on the monitor you can have the visual display and then, you ensure that this solution is complete.

So, once you do that you know in that during that process, if the geometry is simple it may take less time, but then in most of the cases it will take a large amount of time; maybe sometimes the convergence rate is very small or it is you know showing abnormal behavior.

So, we have to choose the relaxation parameters also. Under relaxation parameters in that case that will be basically changing that how much you know taking you are you know how much in what way you are going to incorporate those changes for getting a proper meaningful result from the earlier you know iteration value and that to the next iteration. So, that way you know these relaxation parameters are being used.

So, that way this ensures that equation is properly solved and converged. So, then, once you ensure that it is completely you know you are getting a converge solution.

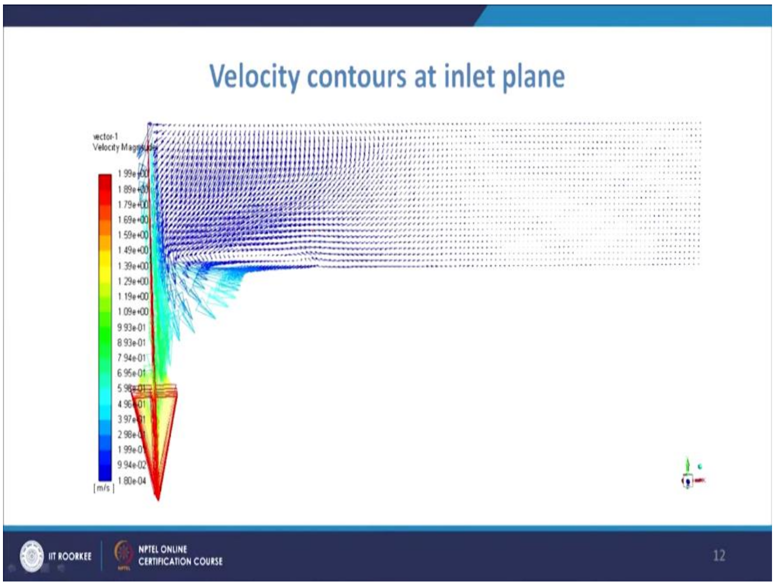
(Refer Slide Time: 21:55)



Then, you go to get the results. You can have a the temperature contours as you see and although, it is showing 1.87 only; 1870 which is basically the inlet temperature. But if you take the range that it will show what it tells that adhere the temperature is maximum and then, you see that on the walls how it is slowly varying and here, it is minimum temperature on these corners and it is decreasing as you see on the top surface.

And on all the walls you can have the temperature contours that will give you the idea that how the temperature is changing you know all across the you know domain. Then, the more important is to know the flow behavior of the steel inside the tundish.

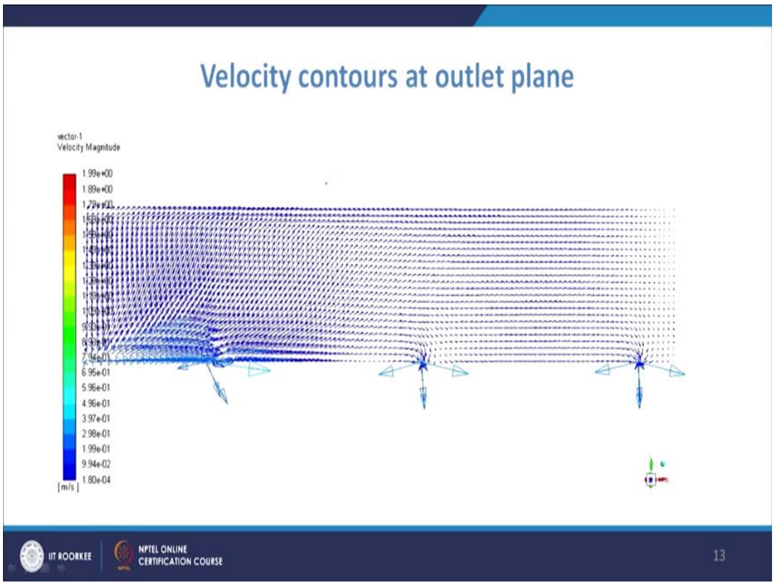
(Refer Slide Time: 22:55)



So, as you see these are the velocity contours at the inlet plane and this is the velocity if you look at the inlet plane as you see this is the inlet vertical plate. So, as you see that you have the flow which is coming on the coming in the bottom direction and then, it is moving from here. So, that way and it is moving through here and then.

So, basically it is a the convective contour is so on. So,, but then this way it is coming and then it is going inside you know through that domain, so, velocity is high and although it is less on this side.

(Refer Slide Time: 23:41)

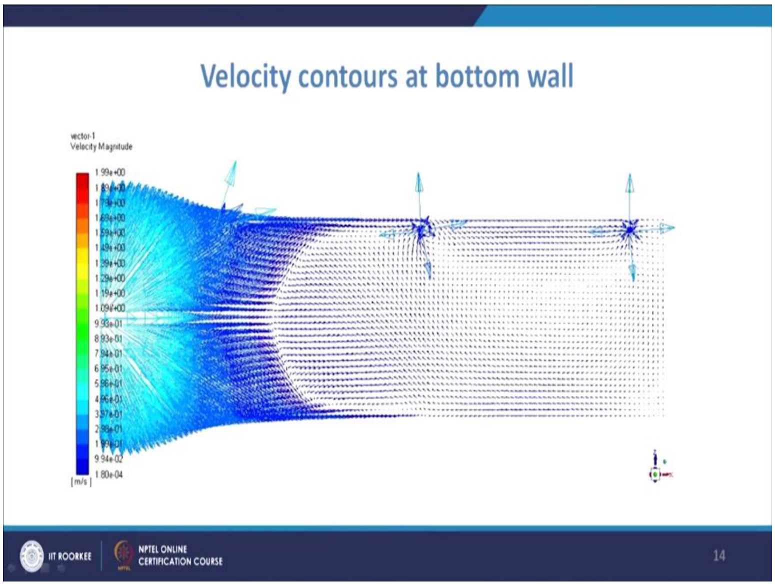


Now, if you take that outlet plane. So, what you see that the metal you know now from here as you see this is towards the you know near the inlet and once it has come, then the liquid metal has the tendency to move up. So, this way it is moving and they are coming and go passing through the outlet.

So, this is how you know you can have the this you can see the velocity of the liquid steel and at these different you know locations and the velocity contours can be seen; maybe we can see also later that there may be you know a kind of you know loops also you can find when you use the flow modifier. So, that time its more visible.

But this is how you can see you know that there will be flow and as you see that the liquid is passing through this at a very high velocity here being near to the inlet, it will be passing and then, as the you know it is moving away from the inlet. So, certainly the velocity at which it will be approaching that will be certainly it may changing because as you move away from the inlet in that case the you know velocity will be changing.

(Refer Slide Time: 25:35)

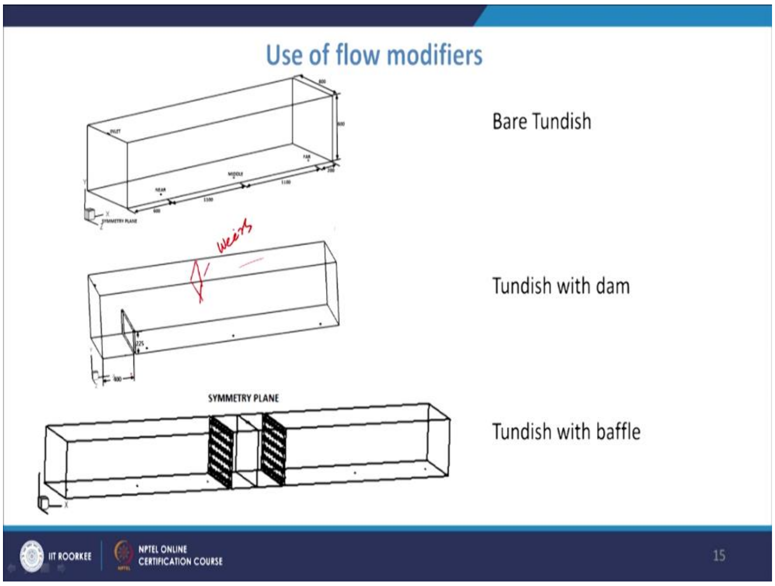


So, that is at the outlet plate. You can have also the you know velocity at the bottom plane. So, as you see the plan view the liquid metal has you know stuck the tundish bottom and then, it is the spreading in all the directions. So, we are having a very high velocity there.

So, it is moving and then as you see that its you know slowly the velocity will be changing. It will be in the surface you know. So, it will be going from here it is you have the first you know. So, you see that considerably very large velocity is coming and then, you will have the metal coming through all these outlets.

So, the velocity can be computed. You can see that where the velocity is high, where the velocity is low. So, these ideas can be obtained from these you know velocity vectors that can be drawn.

(Refer Slide Time: 26:23)



So, the idea is that you know you can very well see the velocity vector you know and you can draw the thermal contours. You can also draw the contours for the turbulence, turbulent kinetic energy which will be indicative of the value of these parameters turbulent kinetic energy, where it is higher. So, where over you have given the velocity higher you likely will you will see that your that values will be more.

So, accordingly because and then you know from these velocities, you can have the idea also that what are those reasons where the velocity is very small. Normally, if the velocity is very small in certain region that may be a probable region for the, dead region for the dead space because the liquid may not the liquid which is coming continuously that may not go into that zone and so that may be the ineffective utilization of the tundish volume.

So, this can be you know seen from these velocity vector values. Now, we can see also that when we use the you know flow modifiers, in that case this velocity vectors change. So, what happens that as we have seen that when we start the you know solution or when we start looking at the velocity vector at the bottom wall or even at the outlet wall, we see that near the outlet it is quickly entering?

So, that is known as short circuiting also many a times. So, if the liquid metal will directly go into that then the height, the high temperature steel will not go into other zones and in that case other zones may experience smaller temperature. So, many a times what we do is we are to alter the flow modifier, I mean flow configuration inside. We use the flow modifiers.

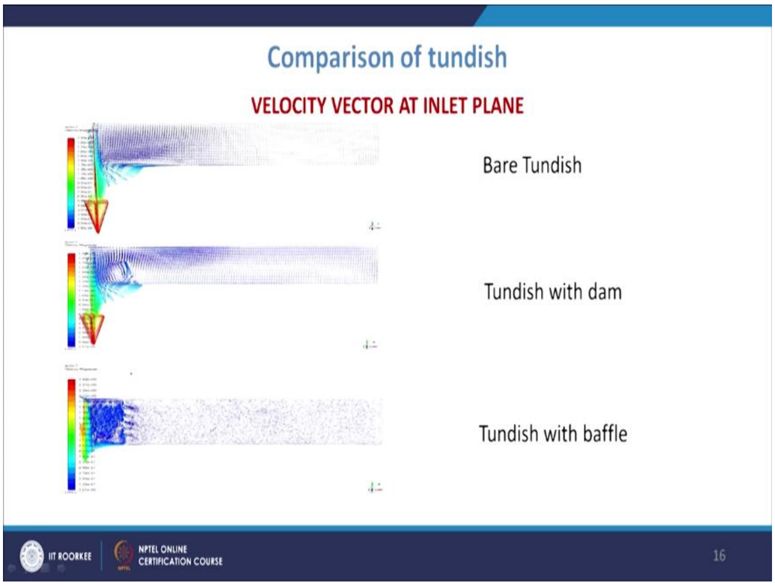
Now, these flow modifiers may be in the shape of a dam which is placed on the bottom wall, you know which is you know it is one position is at the bottom wall at the bottom you know z equal to 0, it will have certain height. Suppose, in this case it has a height up to 225 mm.

So, we may have the a placement of dam just to ensure that the liquid metal which will be striking then after that it will be deflected and it will go inside and then, it will have a loop. So, it will go into other zones and the high temperature steel which is coming, it will be making all the zones you know also active. So, for that we use the you know dams. We also use many a times the weirs. So, weirs our basically used from here.

So, we use from the top at certain you know of certain height. So, that way, we can also use the weirs in these cases and then, we also use the baffles. Baffles will have the you know limited you know outlet area in that whole domain. So, through that the metal will pass. So, so dams weirs baffles and even the advanced pouring boxes are, now it is designed with the boxes are near I mean on the bottom wall and in just below the inlet.

So, they are there to see that what is that they are change on the flow alteration. So, it is a bare tundish; tundish with dam and tundish with baffle.

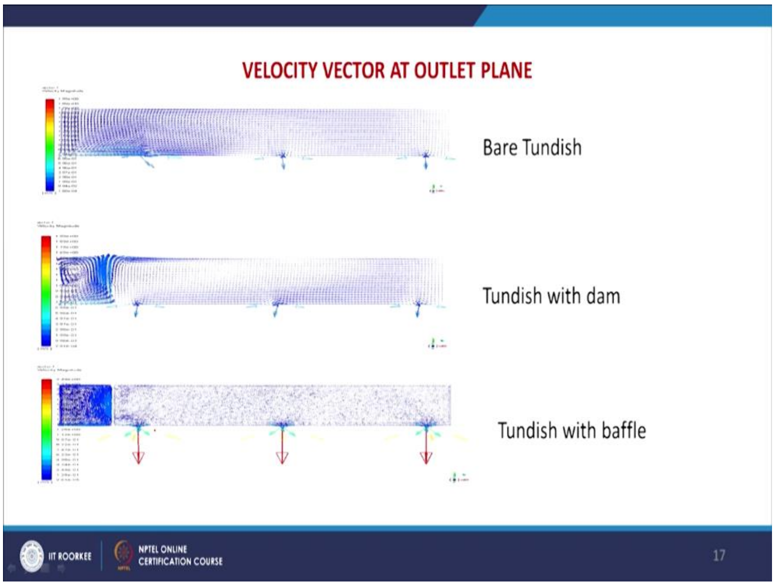
(Refer Slide Time: 30:19)



So, if you take the velocity vector, they you will see that there will be changes in the velocity vector. We will have as you see that being the bare tundish, you will have such kind of estimator; whereas, if you use the dam in that case the metal goes up and then you can see that these higher velocities seen on this side. So, you will have higher velocity. As well as if you use the baffles you can see that it is how these are altering. So, basically it will be you know altering the flow pattern and then the.

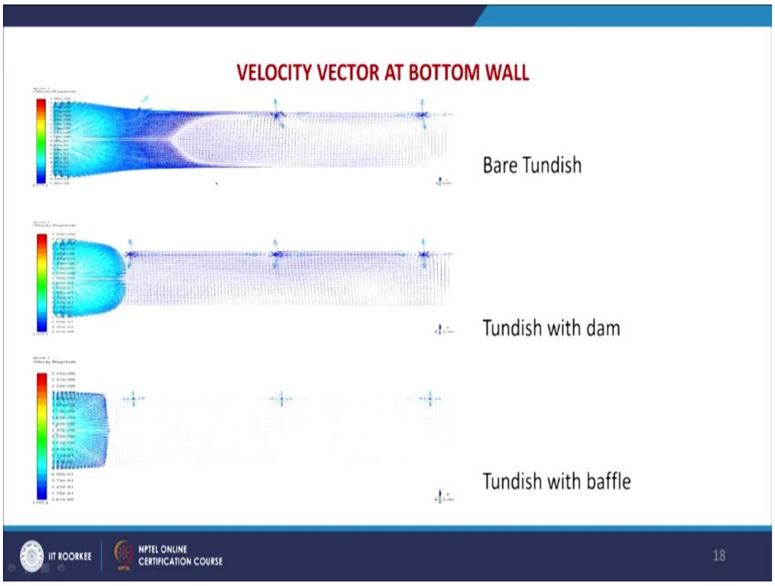
So, they may have there may be different you know purposes for which the tundish is used. Many a times we need you know that it should mix thoroughly; many a times we need a quiescent flow quite flow or so. So, depending upon the situation we can use these different you know flow configurations.

(Refer Slide Time: 31:15)



So, if you look at the outlet plane, you know velocity vector also, there we see that we see there are considerable changes in the you know velocity vector which we see you know in this case, you have the you know this way the flow is going the top surface directed; whereas, in this case is bottom surface directed flow is there. So, accordingly you can have the feel of the change in the you know flow configuration.

(Refer Slide Time: 31:51)



You can also see the velocity vector at the bottom wall. So, as you see that in the case of bare tundish its going like this; but in the bottom wall, you are at the bottom you see the higher velocities up to this zone. Whereas, in this case you see that this high velocity zone is confined here because after that it is facing a dam. So, it is going up. So, velocity is increasing in the upper plains, not at the bottom wall. So, as is seen in the case of dam and baffle also.

So, this is about typically about knowing the fluid flow behavior you know in a tundish and if you do that you know this is used even to find the you know different zones inside the tundish like dead zone, you know plug zone, or mixing zone or so. That we will see in our coming lectures that how we compute these mixing parameters and different tundish volumes inside the tundish.

Thank you very much.