

PROBLEM 4 Steady Laminar Flow and heat transfer in a circular cross section pipe (SOFT COPY SUBMISSION: 12.11.2024)

1. Introduction :

Metallurgical performance (i.e. inclusion float out, thermal mixing, slag emulsification and entrainment etc.) of a steelmaking tundish depends strongly on fluid flow. Tundish geometry and flow modifiers (i.e. pouring box, dam etc.) influence melt flow and thereby exert considerable influence on tundish process performance. Incorporating suitable flow modifiers and placing these at strategic locations, flows in a given tundish could be favorably altered. Optimal designs of such flow modifiers however are rarely deduced from plant scale trials. CFD have been generally applied to evolve suitable designs and identify optimal location of flow modifiers in steelmaking tundish system.

2. The problem

To numerically compute steady, turbulent flow in a WATER MODEL tundish and the residence time distribution characteristics.

- **Newtonian, incompressible, steady, three dimensional** flow and heat transfer with constant properties (Density of fluid= 1000 kg/m^3 and viscosity of fluid= 0.001 kg/(ms)).

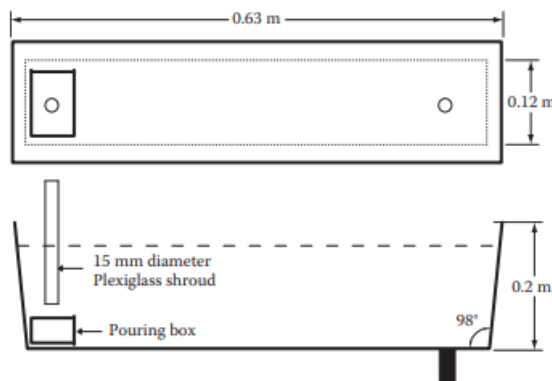


Fig.1 A schematic of the
0.15 scale water model tundish (Pouring box cross sectional area in the model :10cmx10cm)

Water (density = 1000 kg/m^3 and viscosity = 0.001 kg/(ms)) enters the circular cross section pipe at 25 Celsius at an average velocity of 1.273 m/s ($Re_D = 12730$).

Physical Dimensions and Operating Parameters in the 28 ton, Single Strand, Slab Caster Tundish

| Parameters | Numerical Values |
|---|---|
| Tundish length (L): at base and top, m | 4.2841–4.5626 |
| Tundish width (W): at base and top, m | 0.840–1.1326 |
| Melt depth (H), m | 1.042 |
| Velocity at the shroud, m/s | 1.37 |
| Shroud diameter, m | 0.088 |
| Outlet nozzle diameter, m | 0.154 |
| Shroud submergence depth, m | 0.66 |
| Location of the strand and the shroud on the basal plane, m | On the longitudinal central line displaced from the side walls by ~ 0.25 |
| Height of pouring box and dam, m | 0.13 |

IMPLEMENTATION in Fluent

Use Design modeler/CAD etc. and draw the geometry using the volume drawing tool. Create the mesh with quadrilateral surface elements and hexahedral volume elements such that at least 50000 elements (You may choose between 40K and 50K elements). Near the shroud entry region and tundish exit nozzle lay relatively finer grids to capture fluid motion accurately. Declare various surfaces i.e., walls, inlet, outflow and continuum (i.e., fluid) in “Zones” and declare the “SOLVER”. Export the resultant mesh file to Fluent.

Open Fluent, select an appropriate version (i.e., 3D) and read the mesh file. Select the menu “GRID” and carry out a routine grid check-up. Following such, configure the solver (this solves the equation of continuity and momentum) and select **steady and segregated** version. Use standard coefficient k-model with all default prescriptions. Move on to the MATERIALS menu and select water as the bulk fluid. . Set the inlet velocity through the menu “BOUNDARY CONDITIONS and also specify outflow condition at the exit. Use no slip conditions at all walls; deploy wall condition with zero shear stress at free surface of liquid. Declare OPERATING CONDITIONS¹ (gauge pressure etc.). Move on to SOLUTION in the main menu to initialize the problem (with a null velocity field) and specify a maximum number of iterations (say, 5000 or so). Begin the iteration process and solve the problem to arrive at converged solution. First solve the velocity field and then use the converged velocity field to compute tracer dispersion (with inlet mass fraction of neutral density tracer fixed at 1 for 2 seconds of computational time) .

3. Post Processing and Analyses of Computational Results

Once convergence of scalar transport equation is achieved and the computation is terminated, as a routine procedure, write (i.e., save) the CASE and DATA file. Carry out the following post processing and prepare a report on the following suggested line. Submit the same by the deadline to Instructor:

1. Estimate the volume of liquid in the tundish at steady state and hence, calculate theoretical or nominal residence time (= volume of tundish/ volumetric flow rate) numerically.
2. For the given problem, plot concentration (exit) as a function of time and note the minimum break through time (i.e., the time taken by the tracer to reach the probe tip) as well as the time at which the probe registers maximum concentration.
3. Approximate the curve suitably and integrate (upto 2 times the mean residence time τ_{av} . to determine the numerical mean residence time.
4. From 2 and 3 above, calculate the proportions of dead, well mixed and dispersed plug flow volumes
5. Based on steps 2, 3 and 4 draw some inferences on the metallurgical performance of the industrial scale tundish system.

¹ We do not have to specify gravity since numerical flow calculations are carried out with a pressure referenced to local hydrostatic pressure.

