

PROBLEM 3 Steady Laminar Flow and heat transfer in a circular cross section pipe
(SOFT COPY SUBMISSION: 29.10.24)

1. **Introduction** : Bench mark simulations are routinely carried out to assess the adequacy and appropriateness of a CFD procedure. Towards this, numerous test problems and bench mark solutions exist. One such problem, is the steady, forced convective heat transfer and flow in a circular cross section pipe (See any text book on fluid flow and heat transfer or, refer to <https://www.youtube.com/watch?v=jCoA3WLpWhg>). Analytical solution on Nusselt number exists for both constant wall heat flux ($= 4.3$) and constant wall temperature ($= 3.56$) for fully developed flow and heat transfer conditions. These can be used to test the validity of a flow calculation procedure.

2. The problem

To numerically compute steady, laminar velocity and temperature field in circular cross section pipe with constant wall temperature

- **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)).

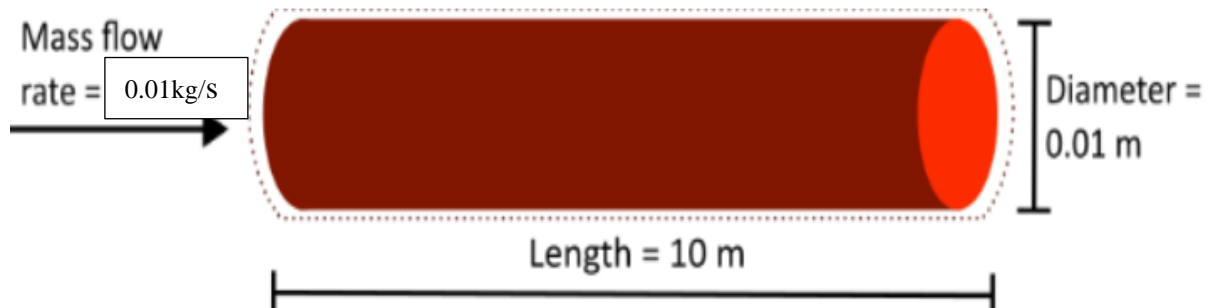


Fig.1 A schematic of the pipe flow and heat transfer problem

Water (density = 1000 kg/m^3 and viscosity = 0.001 kg/(ms)) enters the circular cross section pipe at 25 Celsius at a velocity of 0.1273 m/s ($Re_D = 1273$). The wall of the tube is maintained at a temperature of 75 Celcius (apply thin wall approximation).

3. 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 500000 elements (You may choose between 50K and 60K elements). Ensure relatively finer grid near wall as you have to compute gradient of temperature at wall and hence Nusselt number. 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. Continue with the remaining default settings. Move on to the MATERIALS menu and select water as the bulk fluid. Select the default mode of flow calculation i.e., laminar, and enable energy equation. Set the inlet velocity and temperature through the menu “BOUNDARY CONDITIONS and also specify constant wall

temperature all along 10m length of the pipe. and declare **OPERATING CONDITIONS**¹ (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 heat transfer** (One way coupling philosophy to be adopted).

4. Post Processing and Analyses of Computational Results

Once convergence is reached 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:

- (i) Write down complete formulation of the problem with boundary conditions with the aid of neat sketch labelling dimensions, coordinate systems etc.
- (ii) Note down the number of iterations and computational time for convergence of flow and thermal balance equation. Why flow equation takes relatively more number of iteration to converge? Explain.
- (iii) Calculate Entrance length on the basis of the formula i.e., entrance length = $0.05 \cdot Re \cdot \text{pipe diameter}$ and compare with your numerical estimate. Illustrate your estimation of entrance length from the numerically predicted flow field.
- (iv) Calculate Nusselt number at different locations (consider at least 15 locations) on the wall from $x=0$ to $x=10\text{m}$ (equally spaced) and plot to show the variation as a function of length of the pipe. Does your value at any location come closer to 3.56? comment on your observation.
- (v) Plot cross sectional average temperature as a function of length of the pipe and comment on thermal equilibrium between wall and liquid temperature. What is the exit temperature of fluid in your calculation? REPORT
- (vi) **Write a few comments** on the agreement /disagreement between your computed results and published results.

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