CFD OpenFOAM LAB

Assignments

CL455
Chemical Engineering Department

FOSSEE
IIT Bombay

Funded by Ministry of Education
(MHRD)

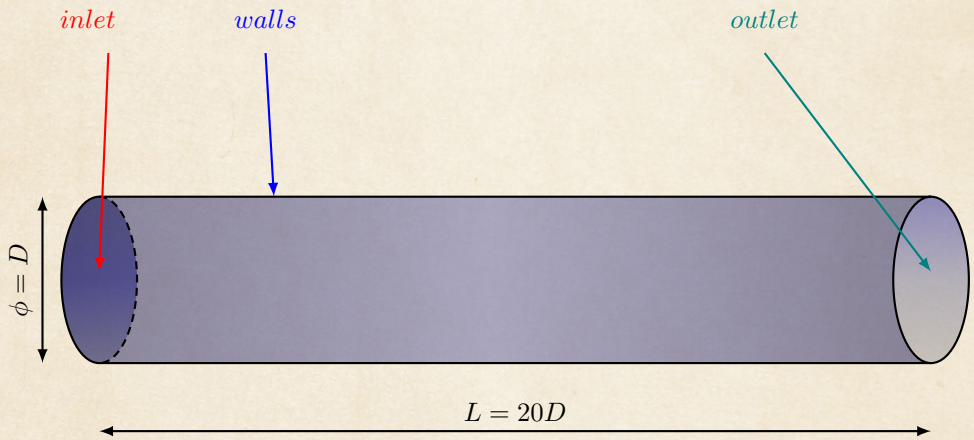
October 21, 2020

Assignment 1: Creating a 3D pipe

Create a pipe having diameter, D and length, 20D using the blockMesh utility in OpenFOAM. The geometry has 3 boundaries, namely, inlet, outlet and the walls of the pipe. Label the boundary patches appropriately. Use the same block configuration and the same numbers of cells and expansion ratios for each block as in tutorial 6: Creating 3D Pipe Geometry and Mesh in OpenFOAM. Take the diameter of the inner cylindrical block (Block 1 from tutorial 6) to be 0.4243D.

D := 2*(last 3 digits of your roll number)/25 mm
For example, if your roll number is 170020072, D = 2*(072)/25 mm = 5.76 mm and L = 20D = 115.2 mm.

Your submission, along with the case files, should contain a snapshot showing both the cross-sectional and axial mesh structure.



Assignment 2: Simulating Flow through a Pipe

Simulate a flow of Reynold number, Re = 500 through the pipe that was created in the previous assignment. Consider the kinematic viscosity to be 1 * 10^{-6} m^2/s. Simulate using a constant time-step of $50D^2$, where D is in meters.

Your submission, along with the case files, should contain a snapshot of the velocity contour at time, t = $8*10^4D^2$ s, where D is in meters.

For example, if your roll number is 170020072, D = 2*(072)/25 = 0.00576 m and t = $8*10^4*0.00576^2$ s = 2.654208 s and time-step = $50*D^2$ = $50*0.00576^2$ = 0.00165888 s

Assuming that the simulation has reached a steady-state, do you think the velocity contour at the outlet represents a fully-developed flow at the end of the simulation? Briefly state why/why not.