

CFD simulation of flow through pipe to validate entrance length and fully developed flow in laminar & turbulent flow

Abstract:

This study aims to validate entrance length for fully developed flow in laminar & turbulent flow through pipe using OpenFOAM CFD software. For laminar case icoFoam & turbulent case pisoFoam solver are use which has capability to solve for transient case of incompressible, laminar/turbulent flow of Newtonian fluids.

Problem statement, models details and initial conditions:

2D pipe was consider for this study which has diameter (d) =0.01m & length (L) =16m. Water is flowing through the pipe having density =1000 kg/m³ and viscosity =0.00089m²/s.

For laminar case Reynolds number consider as 100

For turbulent case Reynolds number consider as 2000

OpenFOAM results for entrance length then compare with analytical solution.

Boundary condition:

Inlet : velocity in “x” direction as per the reynolds number calculation

Outlet: fixed value (101325 Pa)

Walls: noSlip

Other turbulent properties boundary condition calculated using empirical formulae

BlockMeshDict utility is use for meshing.

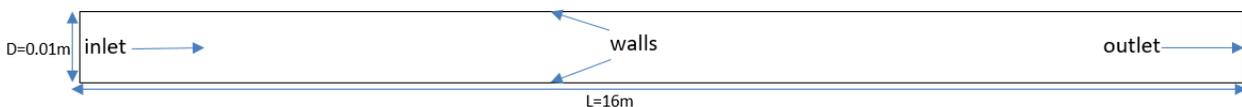


Fig.1. 2D Computational Domain

Manoj Jaylabhdhan Paithane
Department of Mechanical Engineering
MIT Pune