Validation of laminar pipe flow

Dr. Raj Kumar Saini

Ph.D, Indian Institute of Technology, Bombay (IIT Bombay)

M.Tech, Indian Institute of Technology, Madras (IIT Madras)

Email : raj.km.saini@gmail.com

December 7, 2019

Abstract

This case study demonstrates the flow through pipe. 2D case model (geometry and mesh) is made with blockMesh meshing tool/utility. The flow is considered fully developed, steady state and laminar flow. The study is carried out using OpenFOAM-5x. The simulation result for velocity profile is analytically verified and also compared with Ansys-Fluent results.

Keywords: Laminar flow, OpenFOAM, Fluent, Pipe, Reynolds number

1 Introduction

The purpose of this case study is to learn OpenFoam software [1] and to understand two dimensional flow in a pipe for laminar flow regime. In this simulation, a pressure-based finite volume method is used for incompressible flow in a pipe. Fluid dynamics is the science of fluid motion. The simulations are carried out using OpenFoam and Ansys Fluent [1, 2]. The simulation results are validated with analytical result [3]. It is found that OpenFoam is able to accurately simulate the fluid behavior similar as Ansys fluent in each of the above flow regimes.

2 Problem statement

Figure 1 shows the details of the pipe geometry considered in the present study. Pipe with an aspect ratio (L/D) 5 has been taken. Dimensions of the pipe and flow conditions have been shown in Table 1. OpenFOAM solves two-dimensional (2D) axisymmetrical geometry in this case. The Table 1 shows the geometrical parameters and fluid property for the present study.

Table 1: Details of geometry and flow conditions

Geometry details	
Parameters	Value
Length of the pipe (L)	10 cm
Radius of the pipe (R)	1 cm
Diameter of the pipe (D)	2 cm
Fluid Property	
Dynamic viscosity (μ)	0.001 Pa.s

Reynolds number, i.e. $Re = \frac{\rho V D}{\mu} = \frac{1000*0.001*0.02}{0.001}$ = 20.

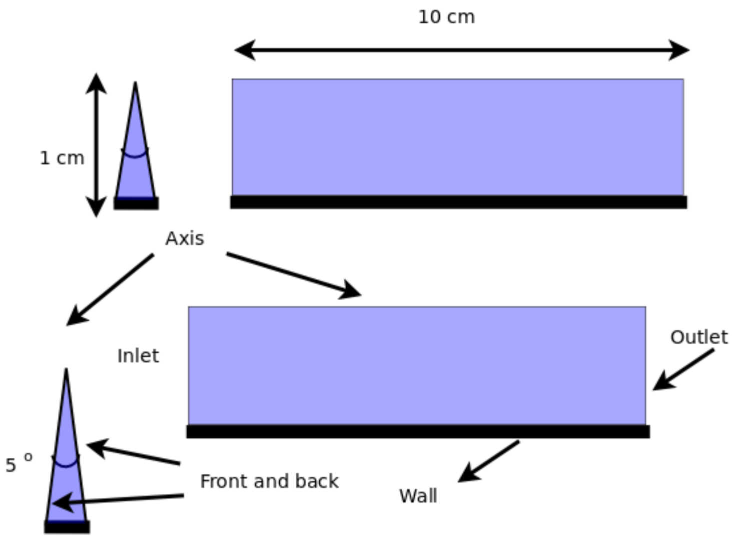


Figure 1: Geometry of pipe

3 Mathematical modeling

3.1 Governing equations

The governing equations for mass and momentum equations for fully developed flow can be written as:

$$\nabla \cdot \vec{U} = 0 \tag{1}$$

$$\frac{d\vec{U}}{dt} + \vec{U}\nabla \cdot \vec{U} = -\nabla\frac{p}{\rho} + \eta\nabla^2\vec{U} \tag{2}$$

Where $\vec{U}$ is the flow velocity, μ and ρ represent dynamic viscosity and density of the fluid considered respectively. η is define as μ/ρ.

3.2 Boundary conditions

Details of boundary name and corresponding boundary conditions are presented in Table 2.

Table 2: Boundary conditions

Boundary Name	Boundary condition
Inlet	Velocity
Outlet	Pressure
Wall	No slip
Axis	symmetry
Front	wedge
Back	wedge

3.3 Analytical solution

The analysical solution for fully developed flow through in a pipe varies with axial position as described in Equation 3 [3].

$$V\left(r\right) = \frac{V_{max}}{2} \times \left[1 - \left(\frac{r}{R}\right)^2\right] \tag{3}$$

4 Simulation procedure

This case deals with two-dimensional laminar simulation of fully developed pipe flow. First step in setting up of an OpenFoam case is to copy to present working directory. We need to set all require input parameters before starting the simulation. Mesh generation and implementation of boundary conditions are adopted from a base cavity tutorial (OpenFOAM/ OpenFOAM-5.x /tutorials/incompressible/icoFoam/cavity). This study is considered with steady state, laminar case. The solver settings are imported from pitzDaily tutorial (OpenFOAM-5.x/tutorials/incompressible/simpleFoam/pitzDaily/) from **simpleFOAM**.

4.1 Creating geometry and mesh

- Geometry for the present problem is considered 2D axisymmetrical domain. The geometry and mesh are generated by using the 'blockMeshDict' file, which is available in 'system' folder in user directory (∼/case/system).
- Suitable modifications for geometry (length and radius of pipe) have been done in the 'blockMeshDict' file in order to generate geometry for the present problem.
- Mesh can be generated by using the command 'blockMesh' in the terminal. Figure 2 shows the isometric view of the generated mesh.

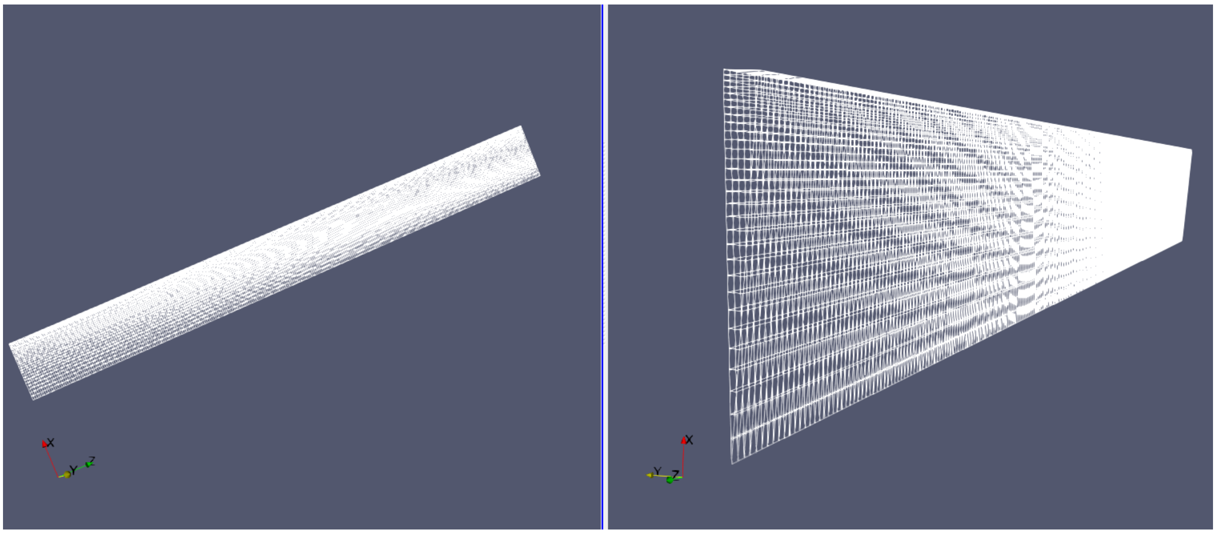


Figure 2: Isometric view of mesh

- All modifications for mesh with proper boundary condition with defining inlet and outlet is to be done as velocity patch and its neighbor patch (∼/case/constant/polyMesh/boundary).

4.2 Setting boundary conditions (BC)/Initial conditions (IC)

Files present in '0' folder (∼/casedirectory/0) has been kept 'p' and 'U' files for laminar and steady flow. Boundaries are assigned and added six boundaries of present case in both the files, i.e. 'p' and 'U'. Details of the boundary conditions are listed in Table 3.

Table 3: Details of boundary conditions

Boundary	p	U
Inlet	zeroGradient	fixedValue (0.001 m/s)
Outlet	fixedValue	zeroGradient
Wall	zeroGradient	noSlip
Axis	symmetry	symmetry
Front	wedge	wedge
Back	wedge	wedge

4.3 Solver details

Steady state and laminar flow are considered in the present study. Laminar flow model can be applied in the OpenFoam by using 'simulationType' option in the 'turbulence Properties' file in constant folder.

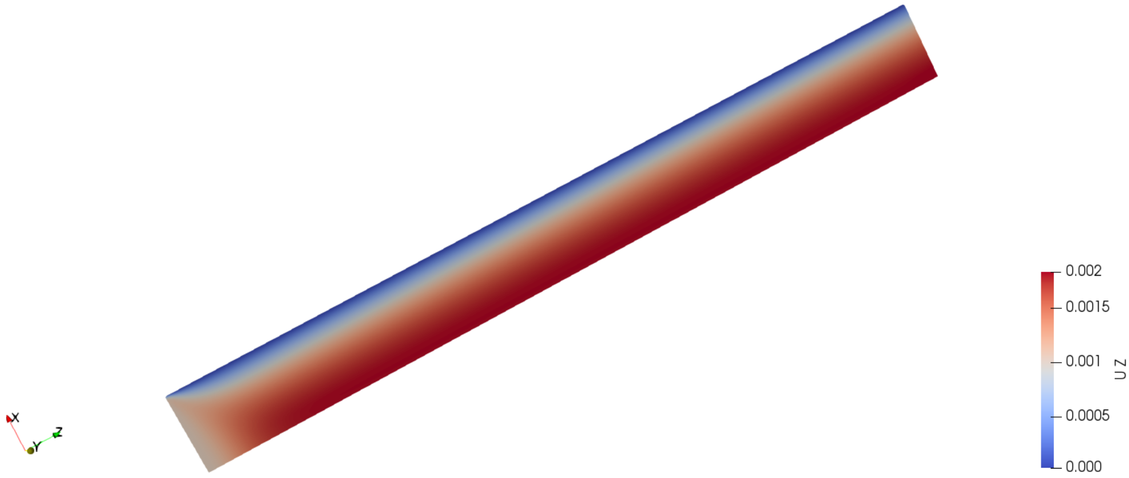
The 'simulationType' option can be used as 'laminar'. In order to run steady state simulations, controDic, fvSchemes and fvSolution files are kept in the case directory folder. '**simpleFOAM**' command executes in terminal to run computations.

5 Validation of the Results

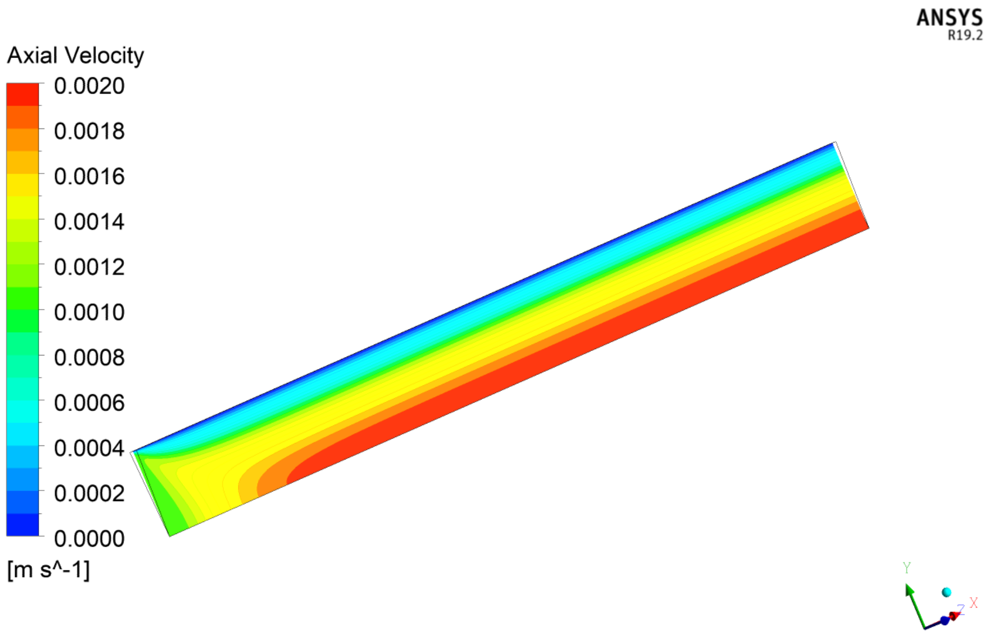
Simulation are performed using OpenFoam and Ansys-Fluent to investigate the flow in a pipe. Simulation results are analyzed with the help of paraFoam software. Figure 3 shows the velocity contour plot for the considered geometry. It can be seen from the Figure that the variation in velocity can not be observed in the flow direction. Analytical result is available in the literature for the axial-velocity profile in a fully developed pipe flow is given as Equation 3.

Figure 4 shows the comparison of simulated values of axial-velocity with the analytical results. This results indicate that the simulated results (OpenFoam and Fluent) are in well agreement with the analytical results.

Results are plotted with the help of 'gnuplot' (∼/case/gnuplot/gnuplot.gp) and the commands used in plotting gnuplot is available in the file 'gnuplot.gp' available in the case directory.



(a) OpenFoam



(b) Fluent

Figure 3: Axial-velocity contour plot for the geometry

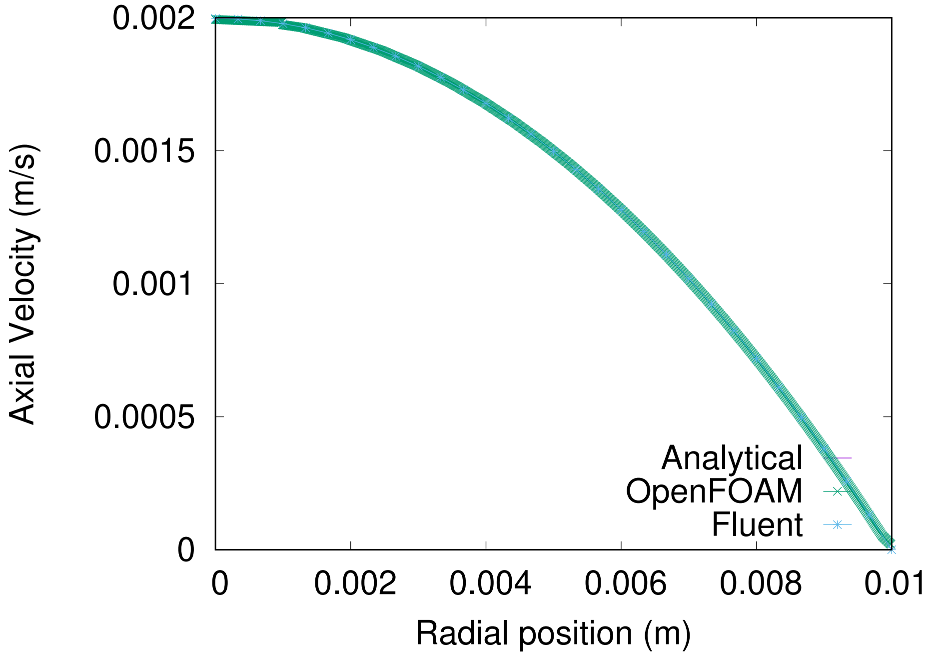


Figure 4: Comparison of simulated results for axial-velocity profile with analytical results

References

[1] C. J. Greenshields, *OpenFOAM: The OpenFOAM Foundation. User Guide Version 5*. CFD Direct Limited, July. 2017.

[2] ANSYS, *ANSYS (Fluent) Theoty Guide*, ser. Release 19.2, 2019.

[3] P. J. P. Robert W. Fox, Alan T. McDonald, *Introduction to Fluid Mechanics (8th ed)*, 8th ed., 2011.