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

Manoj Jaylabhdhan Paithane, MIT, Pune

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 were used which has a capability to solve for transient case of incompressible, laminar/turbulent flow of newtonian fluids.

Key Words: Laminar & turbulent flow, entrance length, developed flow, velocity profile, OpenFOAM, icoFoam, pisoFoam & paraview.

1. Introduction

This report includes CFD analysis of flow through pipe for laminar & turbulent case to validate its developed velocity profile with an analytical equations.

As we know that fully developed flow occurs when the viscous effects due to the shear stress between the fluid particles and pipe wall create a fully developed velocity profile. In order for this to occur the fluid must travel through a length of a straight pipe. In addition, the velocity of the fluid for a fully developed flow will be at its fastest at the center line of the pipe.

Entrance Length and Fully Developed Flow:

At first the fluid is not fully developed when it enters a pipe. Instead the fluid has to travel a certain distance undisturbed before it becomes fully developed. This is also true when a fluid goes around a curve in the pipe system. This is because the curve will disrupt the velocity profile of the fluid. As a result, it will need to travel a certain distance in a straight pipe to become fully developed again. Refer equation 1 to calculate the entrance length for laminar flow, and equation 2 to calculate the entrance length for turbulent flow.

$$\frac{L}{D} = 0.06 \times Re_d$$

……….Eq.1

$$\frac{L}{D} = 4.4 \times (Re_d)^{\left(\frac{1}{6}\right)}$$

……….Eq.2

Where,

L = Entrance Length

D = Pipe Diameter

Re_d = Reynolds Number

Fully developed laminar flow, each fluid particle moves at a constant axial velocity along a streamline and the velocity profile u(r) remains unchanged in the flow direction. There is no motion in the radial direction, and thus the velocity component in the direction normal to flow is everywhere zero. There is no acceleration since the flow is steady and fully developed.

Equation 3 ***[1]*** shows the analytical expression for velocity profile in laminar case

$$(U_r) = 2 \times U_{avg} \times \left(1 - \frac{r^2}{R^2}\right)$$

…......……Eq.3

Where,
U_r= radial velocity (m/s)
U_{avg}= average velocity (m/s)
r= local radial distance from the center of pipe
R= radius of pipe

Fully developed turbulent flow

The velocity profile in turbulent flow is flatter in the central part of the pipe (i.e. in the turbulent core) than in laminar flow. The flow velocity drops rapidly extremely close to the walls. This is due to the diffusivity of the turbulent flow.

In case of turbulent pipe flow, there are many empirical velocity profiles. The simplest and the best known is the **power-law velocity profile** shown by equation 4 ***[3]***

$$U_{Tr} = U_{Tmax} \times \left(1 - \frac{r}{R}\right)^{\left(\frac{1}{n}\right)}$$

………………..Eq.4

f can be computed explixity form *Re*, for example well–known Blasius formula from equation 5

$$f = (100Re)^{-\frac{1}{5}}$$

……………..Eq. 5

Mean velocity to maximum velocity

$$U_{Tmax} = U_{mean} \times \left(1 + 1.33\sqrt{f}\right)$$

......…………Eq. 6

Where,
The exponent n=7 is a constant whose value depends on the Reynolds number
U_{Tr}= radial velocity (m/s)
U_{Tmax}= max velocity for turbulent flow (m/s)
U_{mean}= mean velocity (m/s)
f=friction factor

2. Geometry & Meshing

The geometry was created using BlockMeshDict utility in OpenFOAM. Analysis was done using 2D axisymmetric geometry having d=0.01m & L=16m as shown in fig.1

OpenFOAM only works on 3D model hence we have consider a small slice of geometry having angle of 4° & given 1 unit thickness cell in angular direction and at the side walls given boundary condition was wedge (which means it will not solve equations in z-direction) in this way we can approach for 2D axisymmetric case.

The expansion ratio of 0.1 was given to the Y direction to capture the boundary layer effect near the wall.

BlockMeshDict utility was modified and used for simulation. The detail description of blockMesh shown as follow which include location of vertices, creation of blocks, division of block and boundary conditions as per the geometry definition.

```
convertToMeters 1;
 vertices
((0 0 0) //0
(0 0.005 3.49e-4) //1
(16 0.005 3.49e-4) //2
(16 0 0) //3
(0 0.005 -3.49e-4) //4
(16 0.005 -3.49e-4) //5);
 blocks
(hex (0 4 1 0 3 5 2 3) (25 1 10000)
simpleGrading (0.1 1 1));
 edges();
boundary
(front{type wedge;faces ((1 0 3 2)); }
 back{type wedge; faces( (4 5 3 0));}
 walls{type wall;faces((1 2 5 4));}
 inlet{type patch;faces ((0 1 4 0));}
 outlet{type patch;faces((3 5 2 3));}
 axis{type empty;faces((0 3 3 0));});
mergePatchPairs ();
```

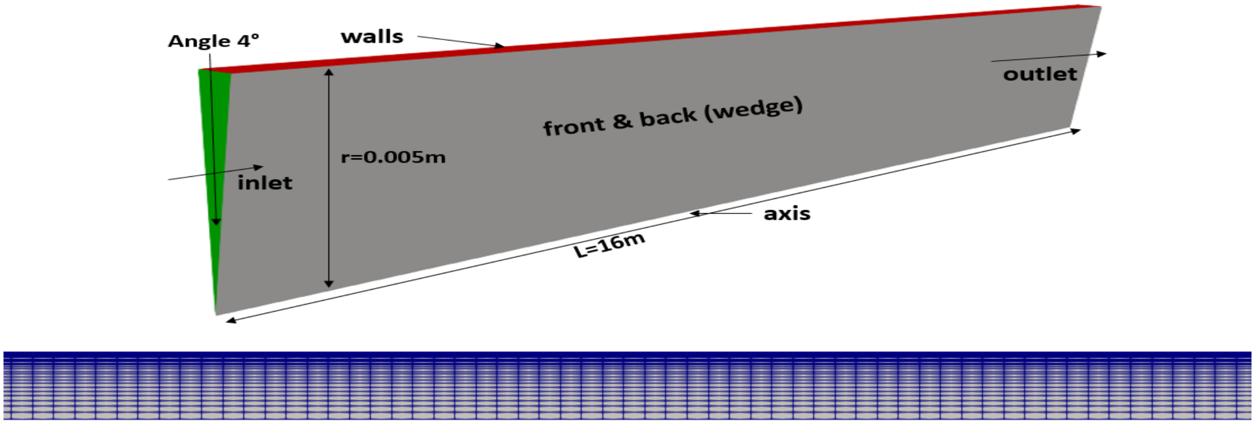

Fig.1. 2D Axisymmetric Computational Domain & Meshing (2.5Lakh elements)

3. Analysis

The CFD analysis of the flow through pipe was done using the software OpenFOAM-v6.
For faster results calculation decomposeParDict utility was used for parallel computing.

4 Boundary condition

In the simulations, the mean velocity inlet boundary condition for the continuous phase is a uniform profile. Turbulence characteristics of the flow i.e. turbulence intensity (I) of 6% is calculated for the inlet flow using Eq. (5). The turbulent kinetic energy k is calculated from U_∞ and I using Eq. (6). The turbulence dissipation rate (ε) is given by Eq. (7) where $C\mu$ is a constant (~ 0.09). The turbulence length scale, l, in this equation is taken as $l = 0.07D_H$ where D_H is the hydraulic diameter of the domain which is equal to the hydraulic diameter of the test section (= 0.01 m).

Fluid used in simulation: water (density 1000 kg/m^3)
Kinematic viscosity = 0.00089 m^2/s

Inlet velocity: 8.9m/s (Laminar case) & 204.7 m/s (turbulent case) with respective Re= 100 & 2300
Initial pressure in entire domain and outlet pressure: 101.325 m^2/s^2
For turbulent case all other boundary conditions are calculated using empirical formulae of Eq. 5, 6 & 7

$$I = 0.16 \times (Re_d)^{-\frac{1}{8}} \text{.........................Eq. 5}$$

$$K = \sqrt{\frac{3}{2}} \times (UI)^2 \text{..............................Eq 6}$$

$$\varepsilon = \frac{(C_\mu)^{\frac{3}{4}}}{l} \times (K)^{\frac{3}{2}} \text{..........................Eq. 7}$$

5 Solver

For the continuous phase flows, the 3D steady RANS equations for conservation of mass & momentum are solved in combination with the k- turbulence model. The PISO algorithm is used and for timeScheme Euler is used, for gradSchemes Gauss linear is used and for pressure-velocity coupling, pressure interpolation is Gauss limitedLinearV 1 and linear discretization schemes is used for both the convection terms and the viscous terms of the equations.

6 Results and discussion

From the analytical equation 1 & 2 we will get entrance length of developed flow in both laminar and turbulent case. The obtained entrance length are as follow

Laminar case (Le) = 0.06m & Turbulent case (Le) = 15.98m

So, at these locations we will extract data for velocity profile in OpenFOAM & compare this results with analytical equation of 3 & 4.

From figure 2 we observed that OpenFOAM predict the exact velocity profile for developed flow in laminar case and it matches well with the analytical solution. Similarly figure 3 shows the prediction of velocity profile for fully developed flow in turbulent case.

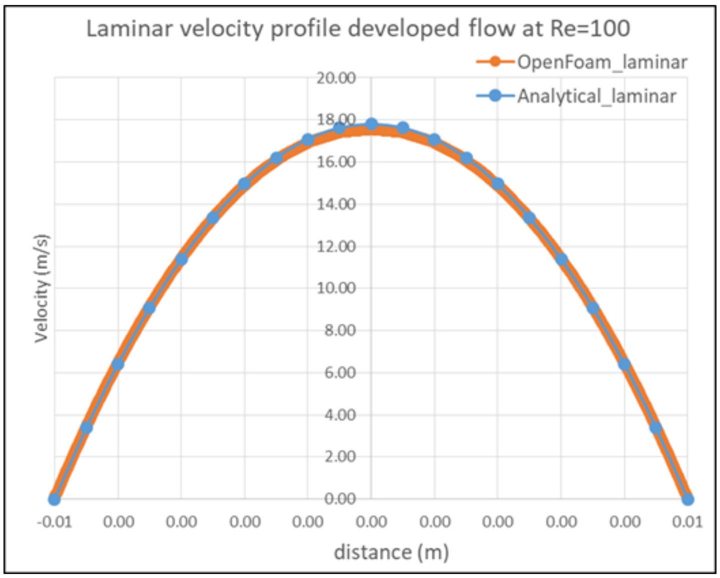

Fig.2. 2D compare laminar developed flow velocity profile at L=0.06m

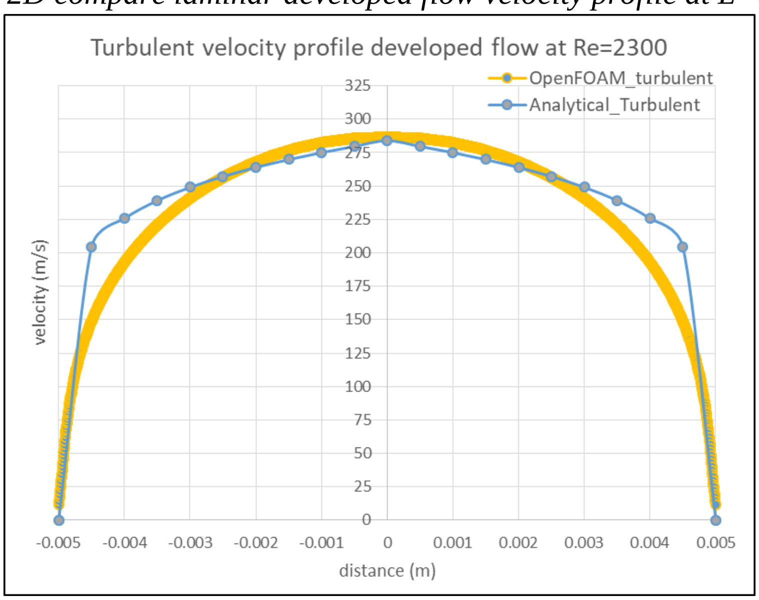

Fig.3. 2D compare turbulent developed flow velocity profile at L=15.98m

Conclusion:

CFD analysis of flow of water through pipe has been studied successfully in the OpenFOAM solver icoFoam & pisoFoam . The simulation produced expected results. The results, especially in the laminar flow matches well with analytical solution & in case of turbulent flow the analytical equation predict good match with OpenFOAM results (here in this case we have consider simple power law equation for turbulent velocity profile).

Command used for simulation:

1. *BlockMesh* : To create axisymmetric geometry
2. *decomposePar*: Parallel computing (division of group of mesh)
3. *mpirun -np 2 icoFoam -parallel > log &* : Solution run in parallel
4. *tail –f log &* : save the log file
5. *reconstructPar* : combines divided meshing to form complete geometry
6. *gnuplot Residuals*: plot residual graph
7. *paraFoam*: post processor (results analysis)

References:

1) Md. Alamin, Md. Ismail Hossain, applying Laminar and Turbulent Flow and measuring Velocity Profile Using MATLAB, Volume 13, Issue 6 Ver. II (Nov. -Dec. 2017), PP 52-5
2) https://en.wikipedia.org/wiki/Entrance_length
3) Turbulent flow, Henryk Kudela