NUMERICAL SIMULATION
OF
LAMINAR AND TURBULENT FLOW
THROUGH THE CIRCULAR PIPE

Submitted by

Monalisha
Department of Civil and Environment Engineering
Birla Institute of Technology, Mesra, Ranchi
24/06/2020

ABSTRACT

The current project is aimed to present the simulation of two-dimensional laminar flow and turbulent flow through the closed pipe using OpenFoam. The aim is to compare the results of different velocities of turbulent flow with each other and to study the dependence of turbulence factors on velocity profile. The aim is to obtain the fully developed flow in the pipe for both turbulent and laminar flow. I have used icoFoam solver for the laminar flow and pimpleFoam for the turbulent flow. I have used RAS (K-epsilon method) for the simulation of turbulence flow. OpenFoam (V 07) and paraview have been used for the project.

Keywords. Laminar, Turbulent, OpenFoam, Fully developed flow, icoFoam, pimpleFoam, entrance length, RAS, k-epsilon.

1. INTRODUCTION

Flow through the pipe is the very common practical problem. Such flow can be seen in industries, houses, sewage carrier pipes etc. So, it is very important to understand the flow to obtain to the better discharge, velocity efficiencies of the flow. Depending on the Reynolds Number, the flow can be laminar or turbulent. The flow is laminar for Reynolds Number lesser than 2300 and it becomes turbulent for Reynolds Number greater than 4000 in the circular pipe. The flow properties of both the types are different. In this project my major aim is to show the fully developed flow. Since, the flow is axis-symmetric, I will perform simulation on the sector of the cross-sectional of the pipe. The length of the pipe I have taken is larger than the length of the entrance length. I have taken water as the fluid for the project and assumed it is an incompressible fluid. To obtain the analytical solution for the fully developed flow, two governing equations (mass and momentum equations) are to be solved. The equations are

1. Continuity Equation

$$\partial/\partial z\ (v_z) = 0 \quad \text{or} \quad v_z = v_z(r)$$

2. Momentum Equation

$$1 + \frac{\mu}{r}\left(\frac{d}{dr}\left(r\frac{d\,v_z}{dr}\right)\right) = -\frac{dp}{dz} = constant < 0$$

3. The final analytical solution obtained

$$u = \frac{R^2}{4\mu}\frac{\partial p}{\partial x}[1 - \frac{r}{R^2}]$$

Here, u is the axial velocity. It is the function of radial distance and it is parabolic in nature for the laminar flow. The velocity profile for the turbulent flow is logarithmic in nature. The average velocity is 0.5 times the maximum velocity in laminar flow and for turbulent flow, the multiplying factor is higher than 0.5 and it varies up to 0.85. In this project I have taken Re= 2100 for the laminar flow and 10^5, 10^6, 10^7 for the turbulent flow. So, the inlet uniform average speed of the flow becomes 0.08m/s for laminar and 4.01m/s (Re = 10^5), 40m/s (Re = 10^6), and 391 m/s (Re = 10^7) for the turbulent flow. The expected analytically maximum velocity after the simulation is,

For Laminar: $V_{max} = 2 * V_{avg}$

Case 01: V_{max} = 2*0.084 = 0.164m/s

For Turbulent: $V_{max} \geq V_{avg}/0.85$

Case 02: $V_{max} \geq$ 4/ (0.85) = 4.7 m/s;

Case 03: $V_{max} \geq$ 40/ (0.85) = 47 m/s;

Case 04: $V_{max} \geq$ 391/ (0.85) = 460 m/s

The pressure drop along the direction should be linearly decreasing for both the laminar and turbulent flow.

Formulae for calculating entrance length,

For Laminar:

$L_e/d = 0.06 * R_e$

For Turbulent:

$L_e/d = 4.4 * R_e^{1/6}$ (Here, d is the diameter of the pipe.)

I have done the simulation for following the cases

Laminar Flow,

S. No	Angle at the centre of the sector	R_e	Velocity (m/s)	Solver used
case 1	4^0	2100	0.084	icoFoam

Table 01

Turbulent Flow

S. No.	Angle at the centre of the sector	Velocity (m/s)	R_e	Solver and Turbulence Model
1	5^0	4.01	10^5	pimpleFoam, RAS->K-epsilon model
2	5^0	40	10^6	pimpleFoam, RAS->K-epsilon model
3	5^0	391	10^7	pimpleFoam, RAS->K-epsilon model

Table 02

2. PROBLEM SETUP

Geometry

The diameter of the pipe is 0.02m for all the simulation cases. The entrance length is calculated using equation 4 and 5 as given in Table 03.

S. No.	Entrance Length of pipe (m)	Provided Length of pipe (m)
Case 1	2.52	3.52
Case 2	0.59	1.3
Case 3	0.88	1.3
Case 4	1.29	1.5

Table 03

Fluid and Flow Characteristics

Flow is assumed to be incompressible, 2-Dimensional, steady for the laminar and transient for turbulent flow. The assumed fluid is water.

The properties of water at 30^0C and 1 atm,

S. No.	Properties	Value
1	Density	996 Kg/m^3
2	Viscosity	$0.799*10^{-3}$ Ns/m^2

Table 04

Initial Conditions

1. Laminar (case 01)

Table 05

S.No.	Locations	Boundary Conditions
1	Inlet	Velocity = 0.084m/s
		Pressure = zeroGradient
2	outlet	Velocity = zeroGrdient
		Pressure = 0 m^2/s^2
3	wall	Velocity = no slip (0m/s)
		Pressure = zeroGadient

2. Turbulent (case 02)

S.No.	Locations	Boundary Conditions
1.	Inlet	Velocity = 4.01m/s
		Pressure = zeroGradient
		K = 0.06 m^2/s^2
		Epsilon = 1.725 m^2/s^3
		Nut = 1.878e^{-4} m^2/s
2.	outlet	Velocity = zeroGradient
		Pressure = 0 m^2/s^2
		K = zeroGradient
		Epsilon = zeroGradient
		Nut = zeroGradient
3.	wall	Velocity = noSlip
		Pressure = zeroGradient
		K = kqRWallFunction
		Epsilon = epsilonWallFunction
		Nut = nutkWallFunction

Table 06

3. Turbulent (case 03)

S.No.	Locations	Boundary Conditions
1.	Inlet	Velocity = 40m/s
		Pressure = zeroGradient
		K = 6 m^2/s^2
		Epsilon = 1724.97 m^2/s^3
		Nut = 1.878e^{-3} m^2/s
2.	outlet	Velocity = zeroGradient
		Pressure = 0 m^2/s^2
		K = zeroGradient
		Epsilon = zeroGradient
		Nut = zeroGradient
3.	wall	Velocity = noSlip
		Pressure= zeroGradient
		K = kqRWallFunction
		Epsilon = epsilonWallFunction
		Nut = nutkWallFunction

Table 07

4. Turbulent (case 04)

S.No.	Locations	Boundary Conditions
1.	Inlet	Velocity = 391m/s
		Pressure=zeroGradient
		K =573.3 m^2/s^2
		Epsilon =1611115.809 m^2/s^3
		Nut =0.0184 m^2/s
2.	outlet	Velocity =zeroGradient
		Pressure=0 m^2/s^2
		K =zeroGradient
		Epsilon =zeroGradient
		Nut =zeroGradient
3.	wall	Velocity =noSlip
		Pressure= zeroGradient
		K = kqRWallFunction
		Epsilon = epsilonWallFunction
		Nut = nutkWallFunction

Table 08

Calculation of the initial conditions for the turbulent flow Case 02

Reynolds Number

Re = (density)*(velocity)*(diameter)/ (viscosity)

Velocity =4.01 m/s.

Turbulent Kinetic Energy

$K = 1.5*(v_{avg} *I)^2$

$K = 0.06\ m^2/s^2$ (v_{avg} =4.01m/s; I=5%)

Epsilon

$\epsilon = ((0.09)^{3/4})*(K^{3/2}/L)$ (Mixing Length (L) = 0.07*length of the pipe)

$\epsilon = 1.725\ m^2/s^3$.

Turbulence kinematic viscosity

$Nut = 0.09*(k)^2/\epsilon$

$Nut = 1.878e^{-4}\ m^2/s$

Meshing

Meshing has been done using openFoam. I have used blockMesh for meshing the pipe.

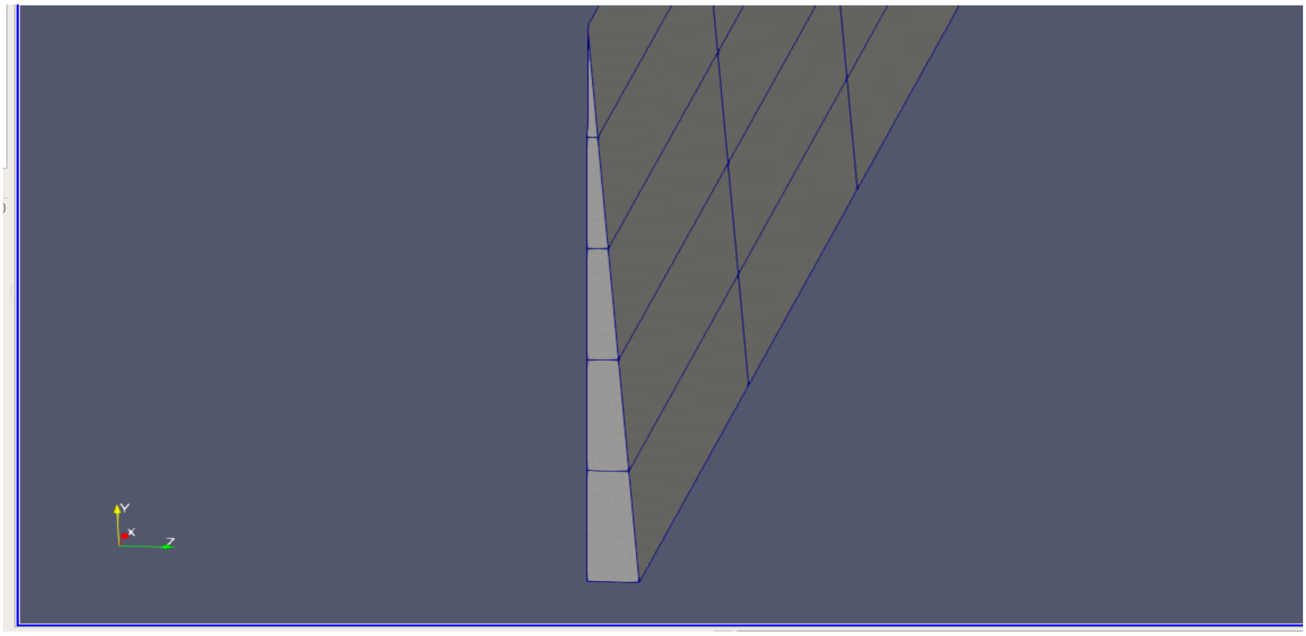


Figure 01: - Meshing

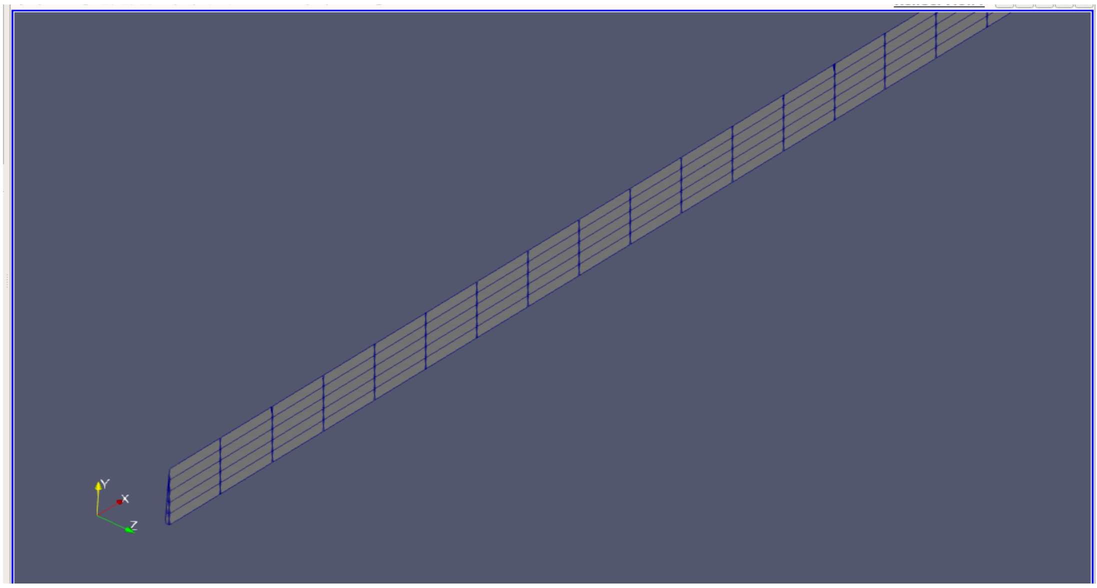


Figure 02: Meshing

Solvers

For laminar flow, icoFoam slover is used to simulate incompressible, steady fully developed flow. For turbulent, pimpleFoam has been used for transient, incompressible flow and RAS (K-epsilon model) has been used for capturing the turbulence characteristics.

3. RESULTS

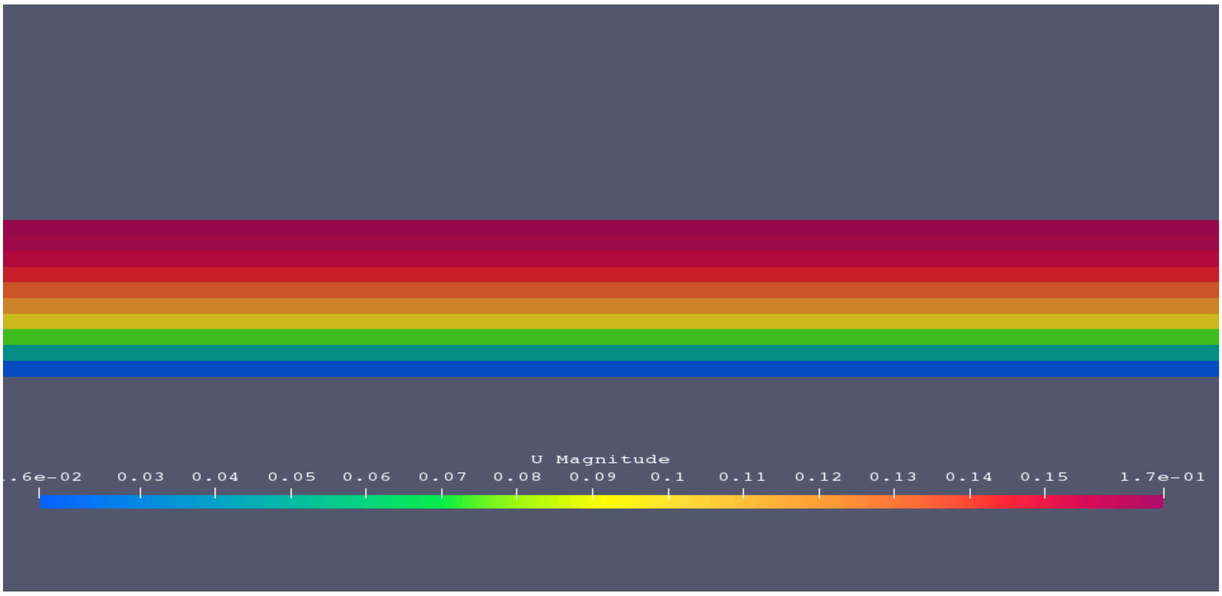
For Laminar

The maximum velocity obtained after simulation is 0.164m/s (figure A, B, C). The fully developed velocity profile has been shown in figure(A). The pressure drop is curved initially but from the end of the entrance length the pressure drops is constant for the given length and it is a straight line.

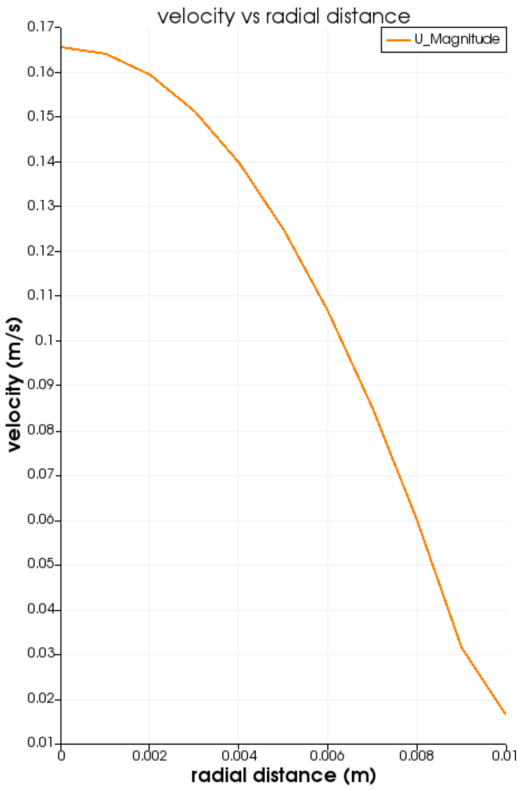
For Turbulent

For case 02, (figure D, E, F), The fully developed flow has been attained very near to the inlet on comparing with the laminar flow. The maximum velocity 4.5 m/s has also been attained after 650 iterations and the profile was not changing for the further iterations. The velocity graph looks more parabolic rather than logarithmic which It should have been analytically. Pressure varies linearly in the fully developed in the turbulent flow. For case 3 (figure G, H, I), The maximum velocity obtained is 44.55m/s. The velocity profile is parabolic shape and the number of iterations required is 200.The value does not change after 200 iterations. The pressure profile is also linear after attaining fully developed flow. For case 03 (figure J, K, L), The maximum velocity is 432m/s. The number of iterations required is 200. The velocity profile is parabolic. The pressure graph is linear after attaining fully developed flow.

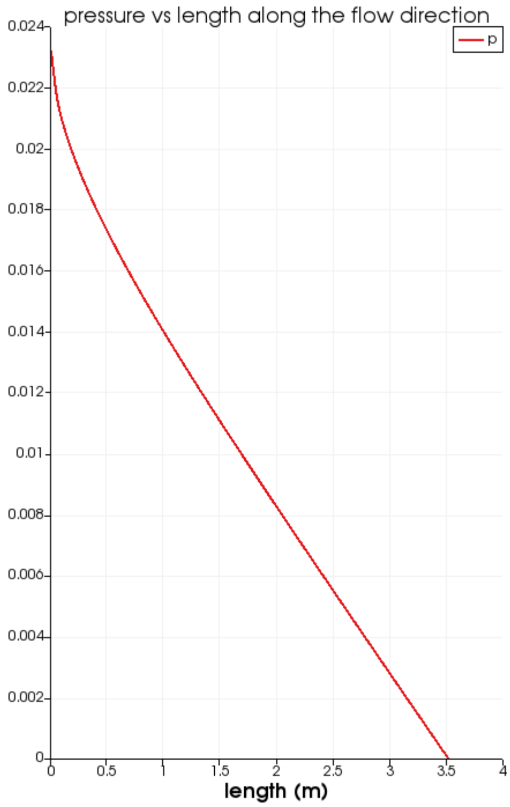
Case 01: Laminar Flow and angle of the sector is 4^0.



(A) Fully developed velocity variation radially

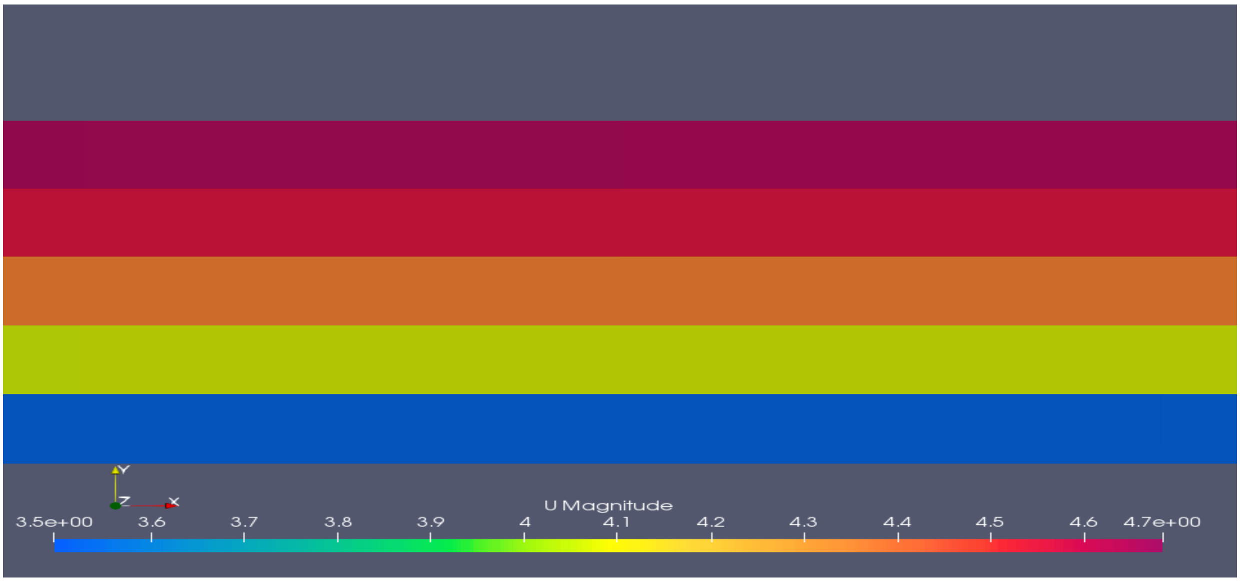


(B) velocity vs radial distance

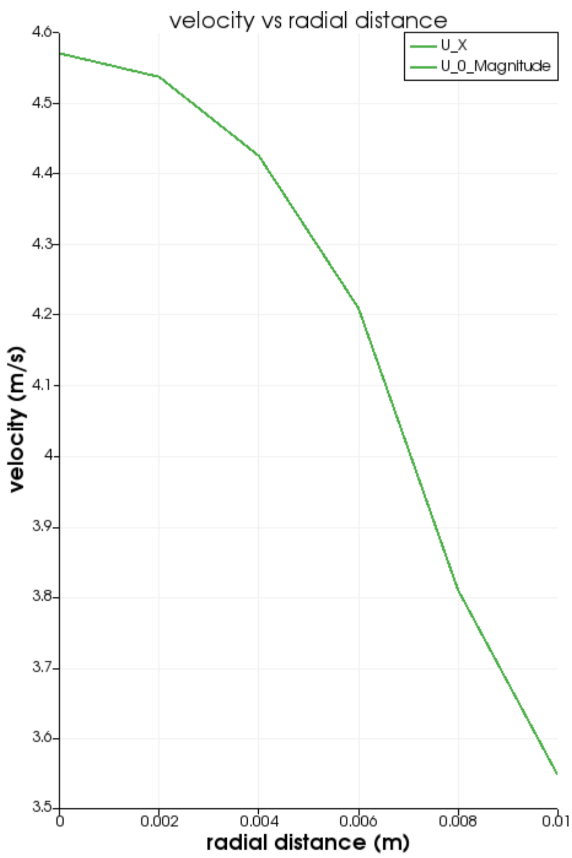


(C) Pressure vs distance

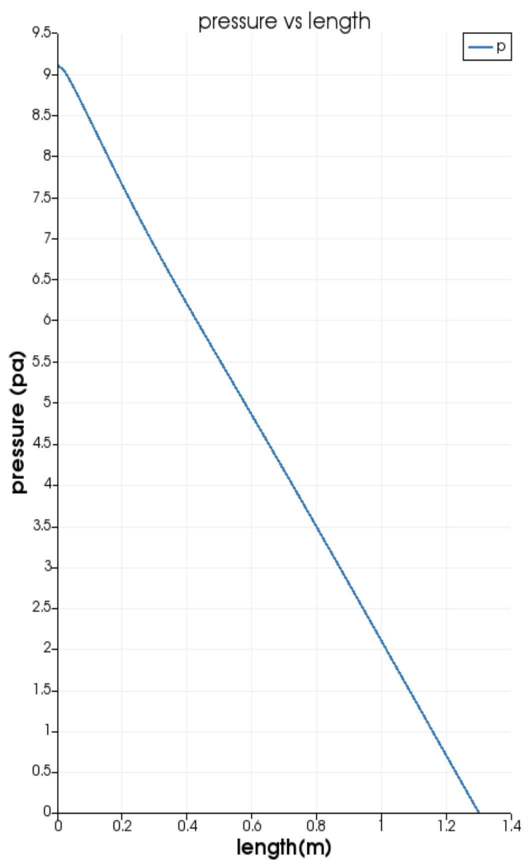
Case 02: Turbulent flow, angle of sector=5^0, velocity =4.01m/s



(D) Fully developed velocity variation radially

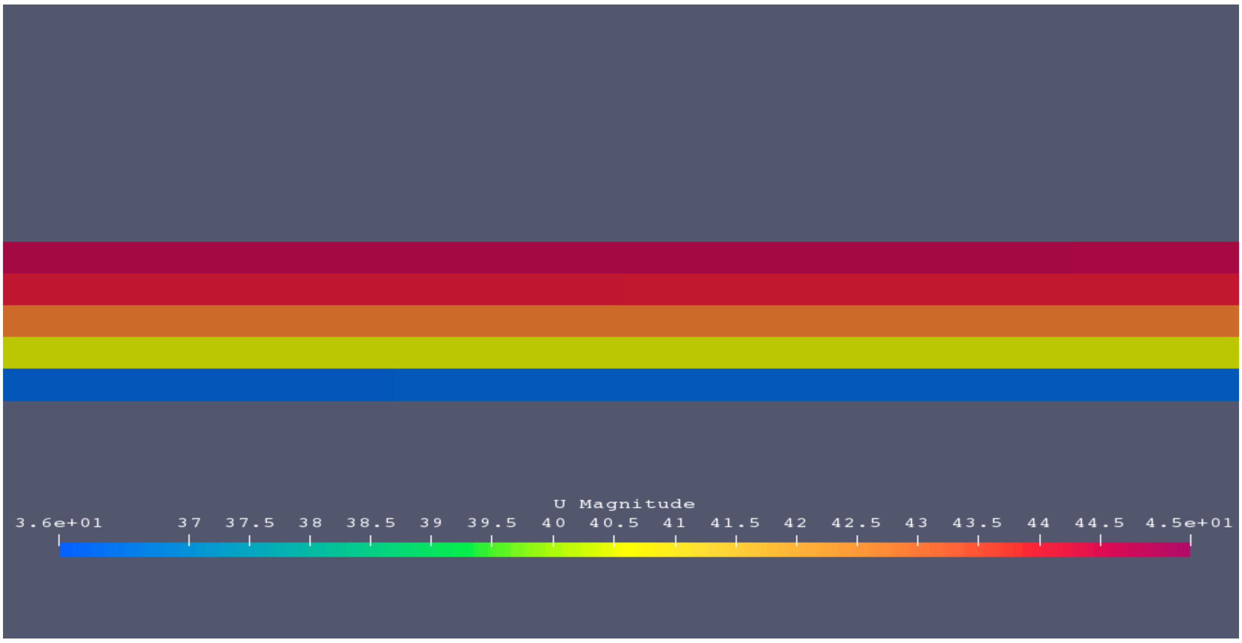


(E) velocity variation radially

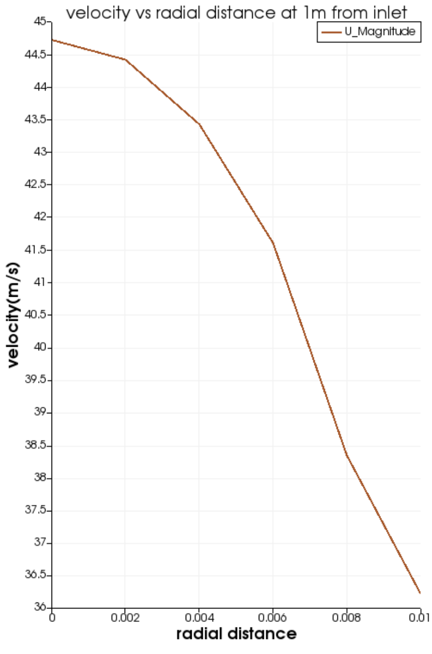


(F) pressure variation

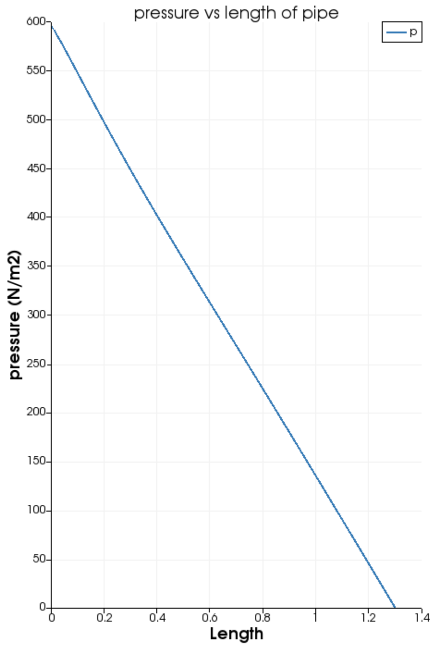
Case 03: Turbulent flow, angle of sector=5^0, velocity =40m/s



(G) Fully developed velocity variation radially

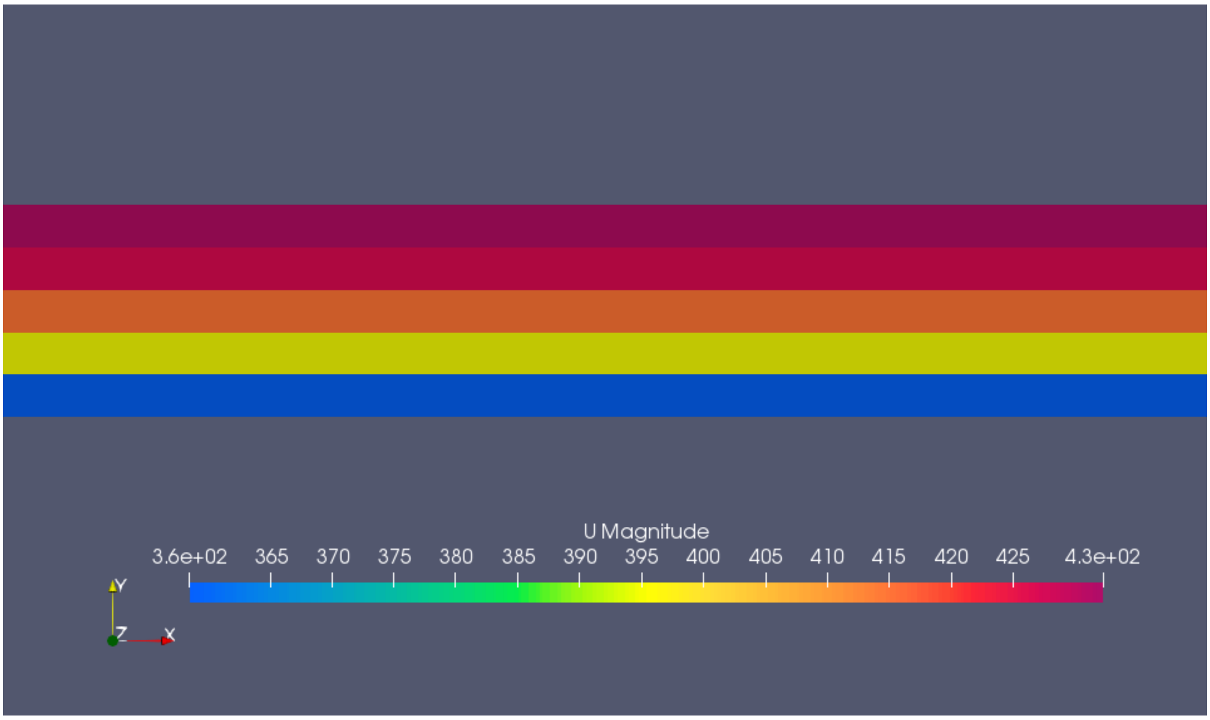


(k) velocity variation radially

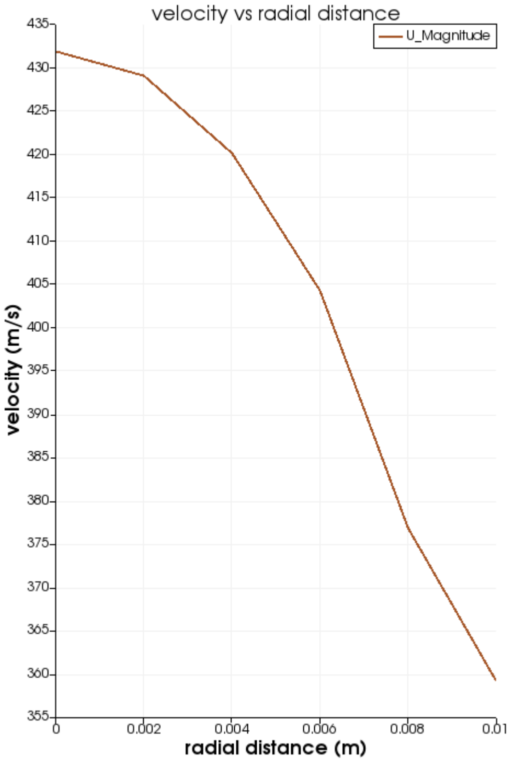


(l) pressure variation along the flow.

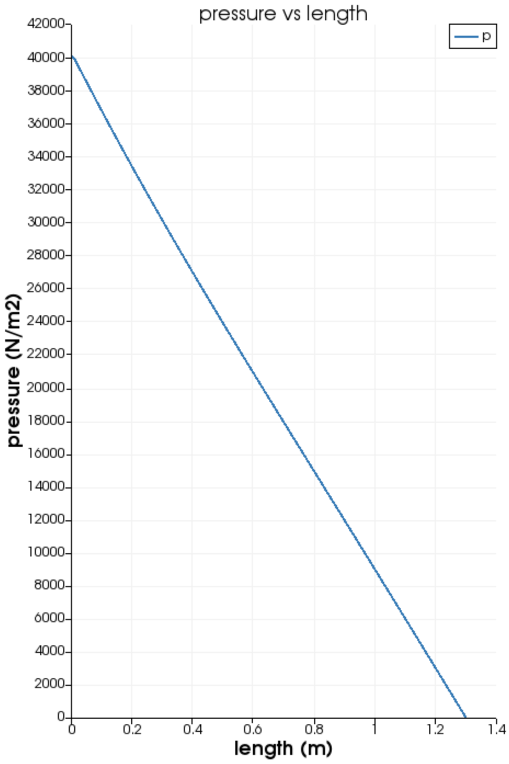
Case 04: Turbulent flow, angle of sector=5^0, velocity =391m/s



(M) Fully developed velocity variation radially



(N) velocity variation radially



(O) pressure variation along the flow.

4. CONCLUSIONS

The simulation of both laminar and turbulent flow using OpenFoam has been done. The use of different solvers and different conditions have been studied. The validation has been done with the analytical solution. The simulation using the given boundary conditions, meshing, solvers and number of iterations can be used to show the correctness of CFD simulation. The obtained maximum velocity is most accurate for the laminar flow. The velocity profile is also parabolic as expected. The maximum velocity obtained is lesser than expected for all the cases. The number of iterations required for the turbulent flow is lesser than that of laminar flow. K is increased from 0.06 to 573 m^2/s^2. The value of maximum velocity has deviate more for the higher K values from the expected value.

References:

1.OpenFOAM Guide

2.Fluid Mechanics, Frank M White

3. https://www.wikipedia.org/

4. https://www.cfd-online.com/Forums/