FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
October 16, 2023

Numerical Simulations of Fluid Flow in Pipe Presenting Effect of Fluid Viscosity and Flow Coefficient (Cv) calculations

Ketan Kulkarni
Savitribai Phule Pune University

Synopsis

This research migration project aims to simulate the numerical simulations of the fluids flow through the pipe using OpenFOAM. The flow simulation uses the solver, simpleFoam to simulate a 2D and 3D pipe flow simulations for three different levels of fluid viscosities such as high viscous (fuel oil), medium viscous (water), and low viscous (alcohol) and compares the results of the simulation against the experimental data analyzed in MATLAB and commercial CFD code Fluent from research paper. The project aims to migrate the study carried out by Baru et. al. Furthermore, this research migration project aims for introducing baffle plate with several holes inside same pipe to control flow and by using superheated steam as fluid. In the first part, this study includes steady state flow simulation using rhoSimpleFoam to calculate flow coefficient (Cv) and compares it with theoretical calculations for Cv. In the second part, transient analysis using rhoPimpleFoam presents fluctuations in axial velocity due to flow restrictions engendered by baffle plate inside pipe. Bejena, Baru & Prabhu S, Venkatesa & Gundaboina, Saikiran. (2021). Computational Fluid Dynamics Simulation and Analysis of Fluid Flow in Pipe: Effect of Fluid Viscosity. Journal of Computational and Theoretical Nanoscience. 18. 805-810. 10.1166/jctn.2021.9680. 1[1]

1 Introduction

The paper focuses on the effect of viscosity on the flow in pipe. The numerical simulations are carried out for three different fluids with high viscosity (fuel oil), medium viscosity (water), and low viscosity (alcohol) by using ANSYS Fluent. The geometry considered is a pipe with 1.5m in length and 0.075m in diameter. Inlet velocity is 0.25 m/s in each case, which gives representation of flow profiles as laminar for fuel oil and turbulent for water and alcohol. Further, MATLAB software is utilized to plot flow profiles using the equations for velocity profile and pressure drop. Thus, the paper compares CFD results with theoretical ones. Now, the whole study is divided into four parts. The study carried out in the paper is migrated in part A. Further, part B consists of study extended to flow control application in pipe with the use of baffle plate of 37 number of holes placed inside the pipe. Steady state CFD analysis is carried out in OpenFOAM V9 with water as a fluid and 1 bar pressure drop as per ISA-75.01.01[2] followed by part C where, steady state analysis with superheated steam at 350°C to calculate pressure drop required for 0.01 kg/s to flow inside the same pipe with the baffle, and to compute Flow coefficient (Cv). Finally, part D involves transient analysis carried out from the already set up flow field of part C to analyze axial velocity variation at three locations in pipe.

2 Governing Equations and Models

Following governing equations are solved in OpenFOAM[3].

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

where ρ is the mass density of the fluid, u is velocity, $\nabla \cdot \mathbf{u}$ is divergence in velocity.

Conservation of momentum:

$$\rho \frac{D\mathbf{u}}{Dt} = \nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{b}$$

where b represents body force per unit mass.

Conservation of energy:

$$\rho \frac{De}{Dt} + \rho \frac{DK}{Dt} = -\nabla \cdot \mathbf{q} + \rho r + \nabla \cdot (\boldsymbol{\sigma} \cdot \mathbf{u}) + \rho \mathbf{b} \cdot \mathbf{u}$$

where the equation describes the rate of change of mechanical and thermal energies for a control volume of material of fixed mass of fluid particles. Thermal energy is described by the specific internal energy e. Mechanical energy is represented by specific kinetic energy K.

Part A: Effect of viscosity on flow in pipe:

To study the effect of viscosity with fuel oil, water and alcohol, simpleFoam solver is used. Realizable k-ε turbulence model is used in momentumTransport file and Newtonian transport model with following values of viscosity is used in transportProperties file. The values of kinematic viscosity are used as given in the paper.

Table 1: Kinematic viscosity values for three fluids considered for part A study

Name of Fluid	**Kinematic viscosity (m^2/s)**
Fuel oil	1.0638e-03
Water	1.0319e-06
Alcohol	1.2471e-06

Part B: Simulation for calculating Flow coefficient (Cv) through baffle inside pipe as per ISA-75.01.01:

This study involves use of simpleFoam solver for incompressible water. As per ISA-75.01.01, water with 1 bar pressure drop and 27°C temperature is used to calculate mass flow rate, which further can be used to calculate Cv. Momentum transport used is as follows:

```
simulationType RAS;
RAS
{
 model          realizableKE;
turbulence      on;
printCoeffs     on;
}
transportModel        Newtonian;
nu                    [0 2 -1 0 0 0 0] 1e-06;
```

Part C: Steady state simulation for calculating Flow coefficient (Cv) through baffle inside pipe for process condition

Superheated steam is widely used in process industry and a challenging task in this study is to control and allow 0.01 kg/s of flow rate through pipe. This study uses steady state compressible rhoSimpleFoam solver. Real gas Peng Robinson model[4] is used as equation of state and Sutherland model[5] is used for viscosity. Thermophysical properties are as shown below.

```
  thermoType
  {
    type                hePsiThermo;
    mixture             pureMixture;
    transport           sutherland;
    thermo              hConst;
    equationOfState     PengRobinsonGas;
    specie              specie;
    energy              sensibleInternalEnergy;
 }
mixture
 {
   specie
    {
     molWeight   18.01534;
    }
    thermodynamics
    {
```

```
      Cp        2135.05; //J/kg
      Hf        0;
    }
    equationOfState
    {
     Tc         647.14; // critical temp K
     Vc           0.0559; //critical volume m^3/kmol
     Pc         2.2064e+07; // Critical pressure Pa
     omega      0.344; //Accentric factor (-)
    }
    transport
    {
      As        2.75e-06;
      Ts        1313.183;
    }
}
```

Part D: Transient simulation for analyzing axial velocity variations through baffle inside pipe for process condition

This case utilizes compressible transient solver rhoPimpleFoam with Peng Robinson equation[4] of state and Sutherland viscosity[5].

3 Simulation Procedure

3.1 Geometry and Mesh

A geometry is created using space-claim modeler and saved it as an STL file to use in OpenFOAM.
Part A:

A geometry used in this study is a pipe of 1.5m length and 0.075m diameter. A blockMesh utility is used to create a single box and then the STL file is meshed using snappyHexMesh utility. Global level 5 and local level 5 refinement is used with 8 inflation layers at boundary. Number of cells are approximately 9 lakhs. Mesh is as shown in a figure below.

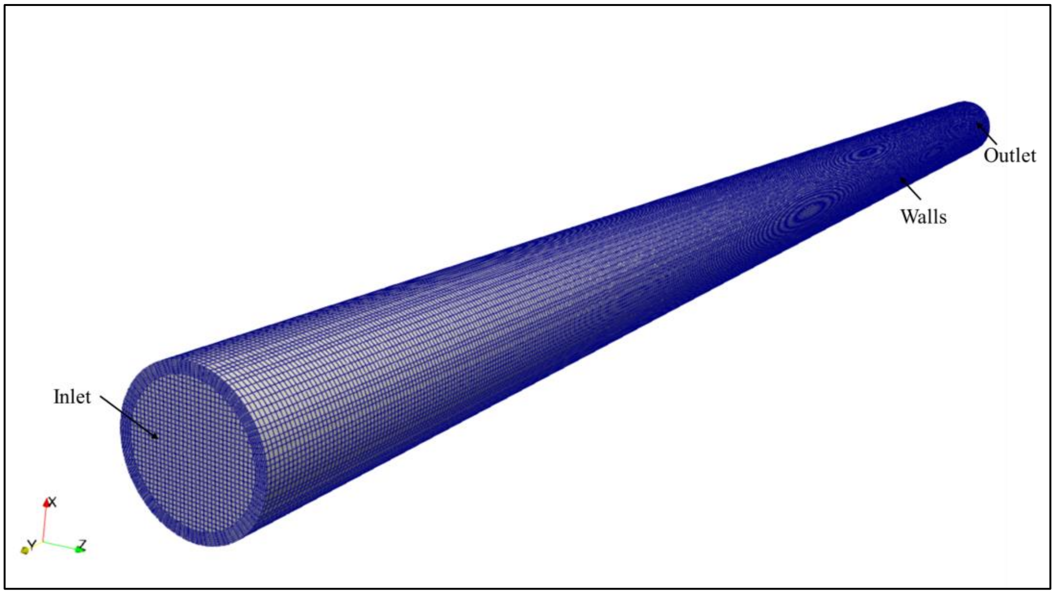

Figure 1: Meshed model for the part A study

Part B:

A geometry used in this study is a pipe of 1.5m length and 0.075m diameter with a baffle of 37 number of holes of 6 mm diameter at the mid-section of pipe. A blockMesh utility is used to create a single box and then the STL file is meshed using snappyHexMesh utility. Global level 5 and local level 5 refinement is used except baffle section, which is meshed with local refinement level 7. Along with this, 3 inflation layers at boundary. Number of cells are approximately 38 lakhs. Mesh is as shown in a figure below.

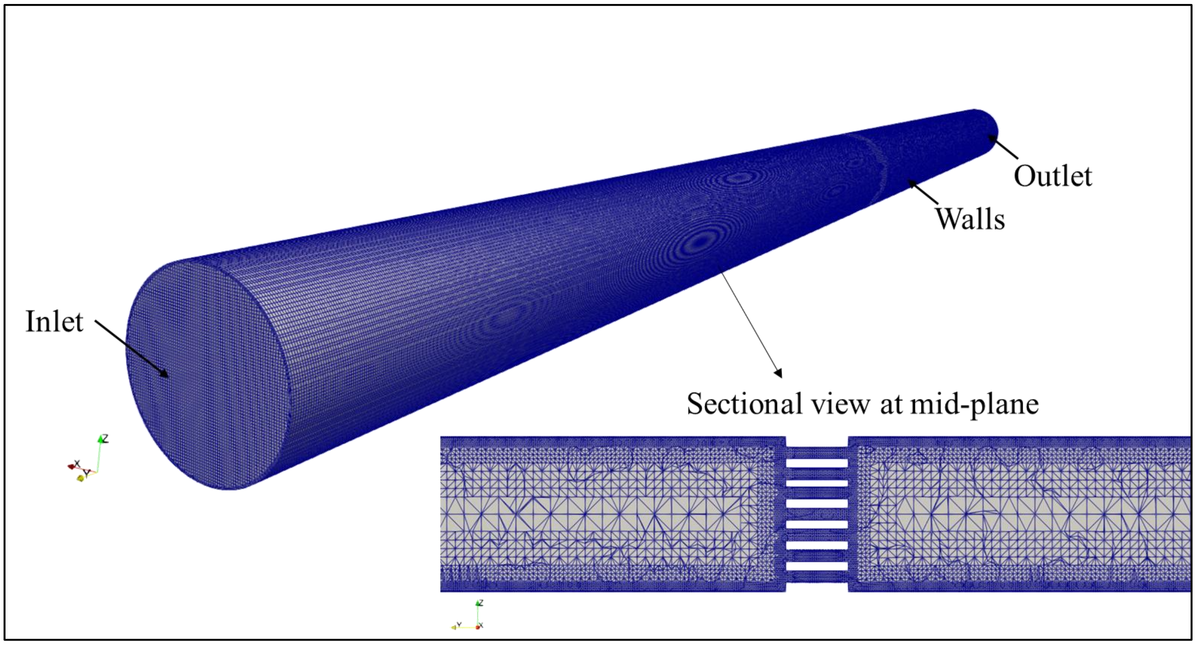

Figure 2: Meshed model for the part B study

Part C and D:

The geometry used in study part C and D is a pipe of 0.48m length and 0.075m diameter with a baffle of 37 number of holes of 6 mm diameter at the mid-section of pipe. A blockMesh utility is used to create a single box and then the STL file is meshed using snappyHexMesh utility. Global level 3 and local level 3 refinement is used. Along with this, 3 inflation layers at boundary. Number of cells are approximately 9 lakhs. Mesh is as shown in a figure below.

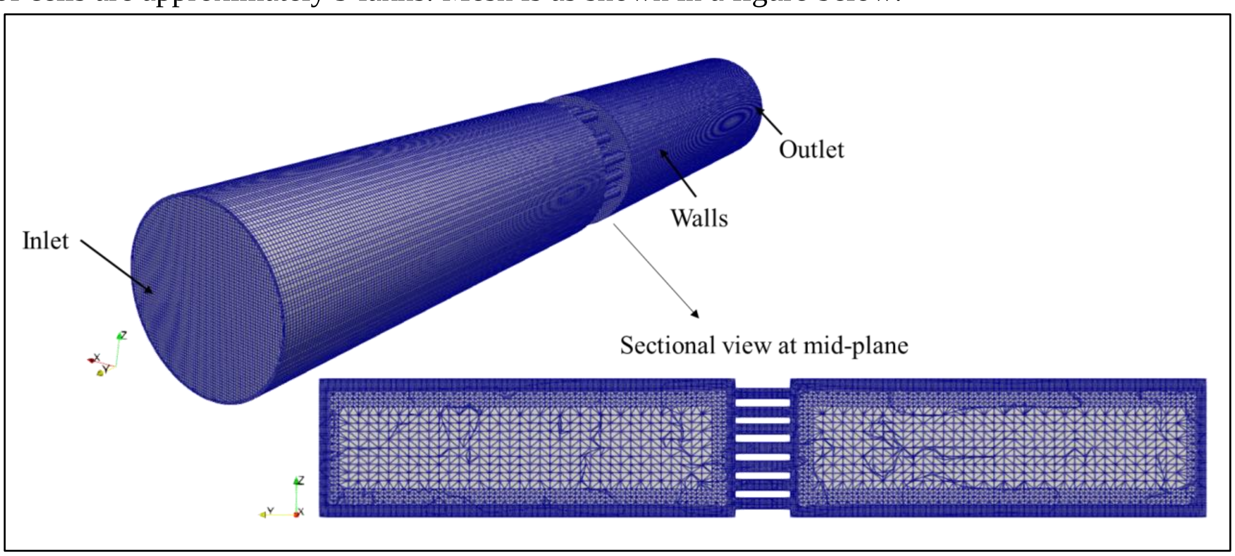

Figure 3: Meshed model for the part C and D study

3.2 Initial and Boundary Conditions

Initial and boundary conditions used for each part of the study are tabulated as follows.

Table 2: Initial and boundary conditions for part A study

Part A			
Initial conditions	Inlet	Outlet	Wall
Pressure, p	zeroGradient	fixedValue uniform 0	zeroGradient
Velocity, u	fixedValue uniform (0 -0.25 0)	zeroGradient	noSlip
Turbulent kinetic energy, k	fixedValue	zeroGradient	kqRWallFunction
Disspation Rate, ε	fixedValue	zeroGradient	epsilonWallFunction
Tuebulent viscosity, nut	calculated	calculated	nutkWallFunction

Table 3: Initial and boundary conditions for part B study

Part B			
Initial conditions	Inlet	Outlet	Wall

Pressure, p	type uniformTotalPressure
	p0
	{
	type table;
	values 2((0 900)(1 1000));
	}
	value uniform 1000;
		fixedValue
		uniform 900
			zeroGradient

Velocity, u	pressureInletOutletVelocity uniform (0 0 0)	inletOutlet uniform (0 0 0)	noSlip
Turbulent kinetic energy, k	turbulentIntensityKineticEnergyInlet	inletOutlet	kqRWallFunction
Disspation Rate, ε	turbulentMixingLengthDissipationRateInlet	inletOutlet	epsilonWallFunction
Tuebulent viscosity, nut	calculated	calculated	nutkWallFunction

Table 4: Initial and boundary conditions for part C and D study

Part C and D			
Initial conditions	Inlet	Outlet	Wall
Pressure, p	zeroGradient	fixedValue uniform 101325	zeroGradient

Velocity, u	flowRateInletVelocity
	massFlowRate
	{
	type constant;
	value 0.01;
	}
	rhoInlet 1.4;
		inletOutlet	noSlip

Temperature, T	fixedValue uniform 623	inletOutlet	zeroGradient

Turbulent kinetic energy, k	turbulentIntensityKineticEnergyInlet	inletOutlet	kqRWallFunction
Disspation Rate, ε	turbulentMixingLengthDissipationRateInlet	inletOutlet	epsilonWallFunction
Tuebulent viscosity, nut	calculated	calculated	nutkWallFunction
kinematic turbulent thermal conductivity	calculated	calculated	compressible::alphatWallFunction

Turbulence characteristics of the flow i.e. turbulence intensity (*I*) is calculated for the inlet flow using Eq. (1). The turbulent kinetic energy (k) is calculated from *U*∞ and *I* using Eq. (2). The turbulence dissipation rate (ε) is given by Eq. (3) where *Cμ* is a constant (~ 0.09). The turbulence length scale, *l*, in this equation is taken as $l = 0.5D_H$ where D_H is the hydraulic diameter of the domain, which is equal to the hydraulic diameter of the pipe.

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

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

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

3.3 Solver

For continuous phase flows, RANS equations for conservation of mass, momentum and energy are solved in combination with the realizable k-epsilon turbulence model. The solvers used in each study are as below.

Table 5: Solver used in each part of the study

Name of the study	Solver used
Part A: Effect of viscosity on flow in pipe	simpleFoam
Part B: Simulation for calculating Flow coefficient (Cv) through baffle inside pipe as per ISA-75.01.01	simpleFoam
Part C: Steady state simulation for calculating Flow coefficient (Cv) through baffle inside pipe for process condition	rhoSimpleFoam
Part D: Transient simulation for analyzing axial velocity variations through baffle inside pipe for process condition	rhoPimpleFoam

4 Results and Discussions

- **Part A:**

The flow profile depicted in the paper and the results obtained from OpenFOAM in part A of the study are compared as below. The comparison holds good for velocity profile measured along the diameter of pipe between the results from the paper and part A study. Refer figure 4 and 5 for velocity flow profiles comparison and figure 6 to 9 for velocity vector representation.

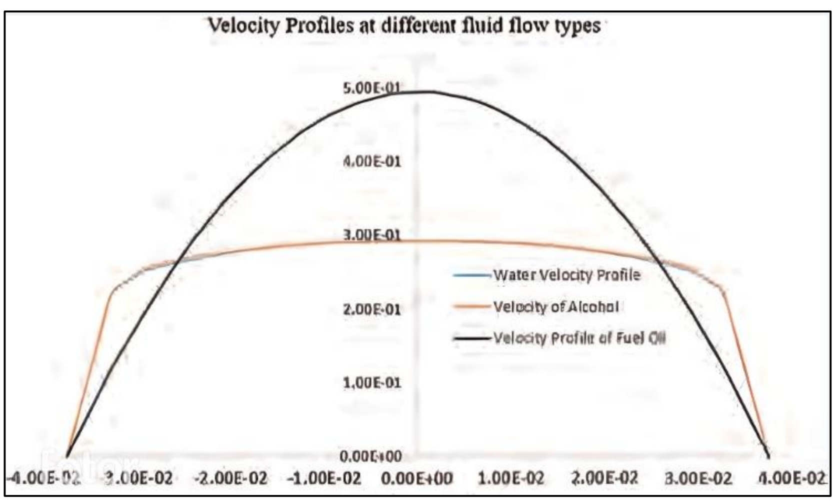

Figure 4: Velocity flow profiles for the three fluids documented in the paper

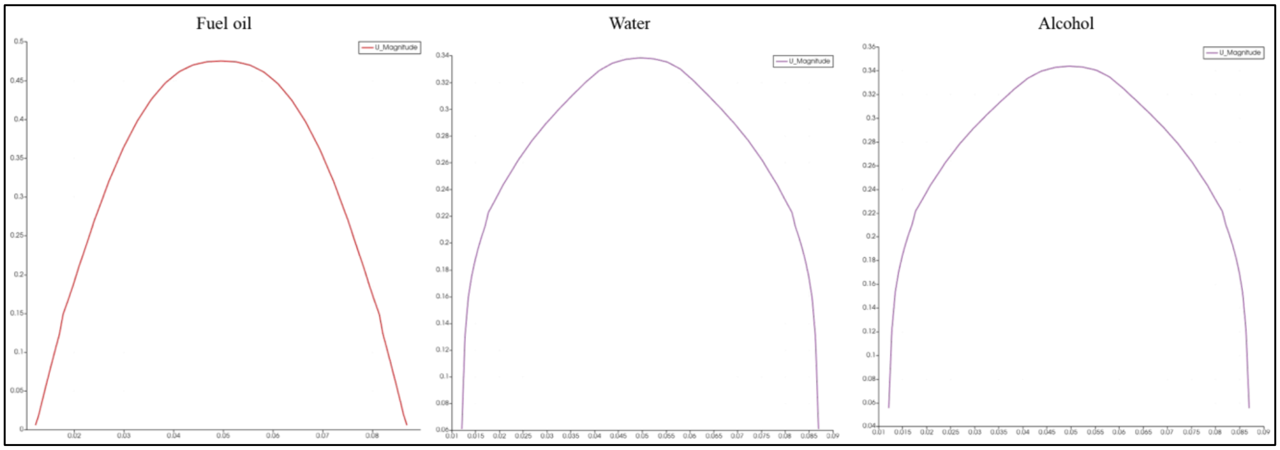

Figure 5: Velocity flow profiles for the three fluids of part A study in openFOAM

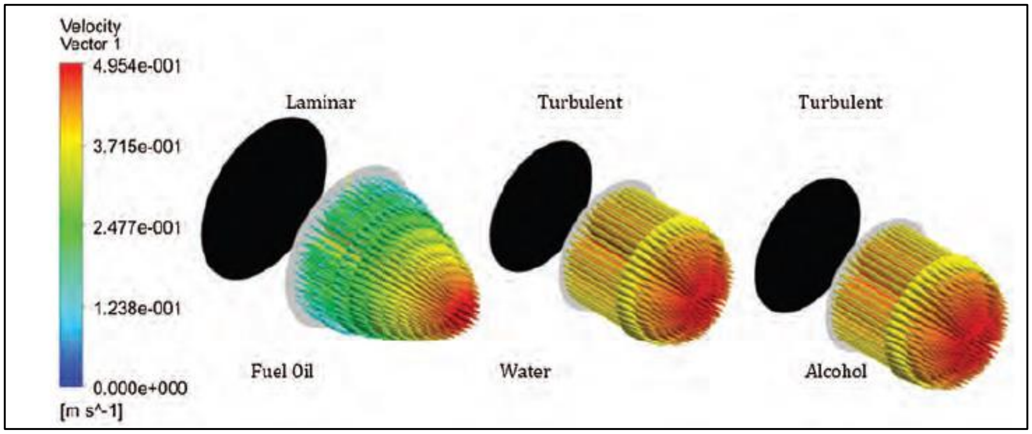

Figure 6: The representation of velocity vector for the three fluids documented in the paper

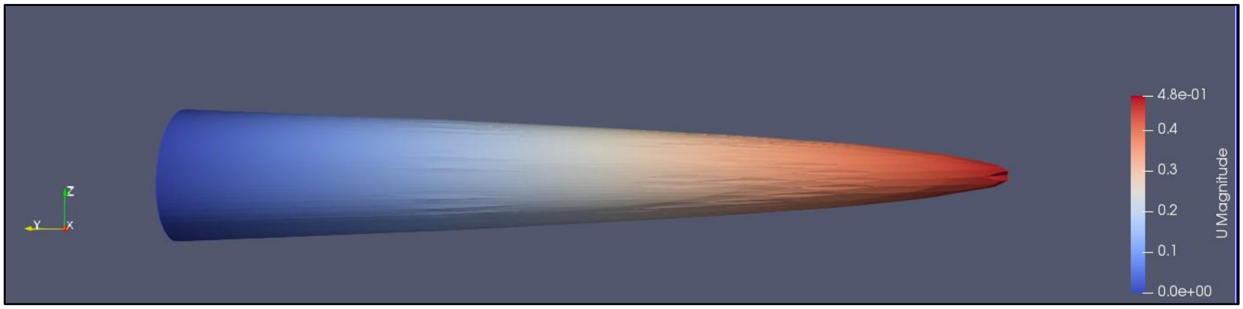

Figure 7: The representation of velocity vector for fuel-oil (laminar flow profile)

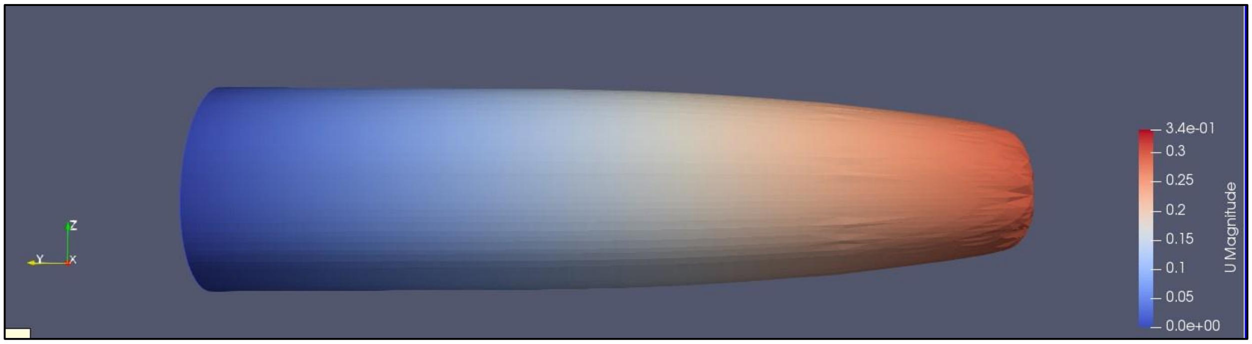

Figure 8: The representation of velocity vector for water (turbulent flow profile)

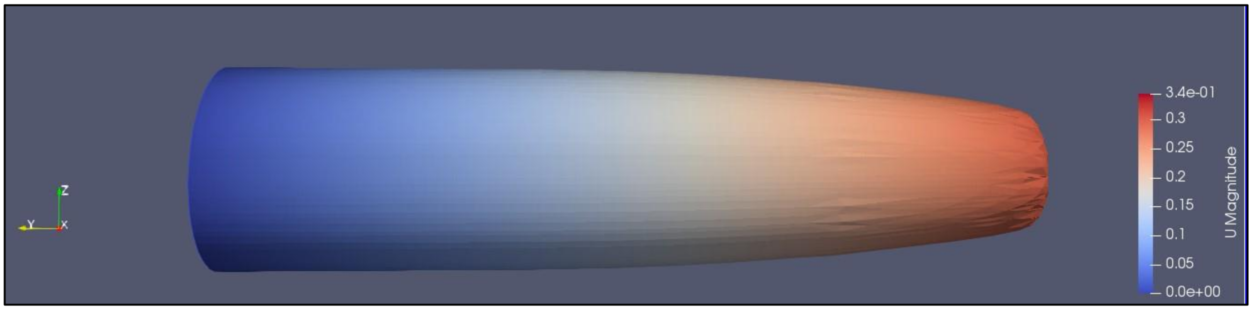

Figure 9: The representation of velocity vector for alcohol (turbulent flow profile)

- **Part B:**

Pressure and velocity contours are shown below to describe water flow inside the pipe with baffle. Figure 9 shows kinematic pressure at inlet and outlet respectively.

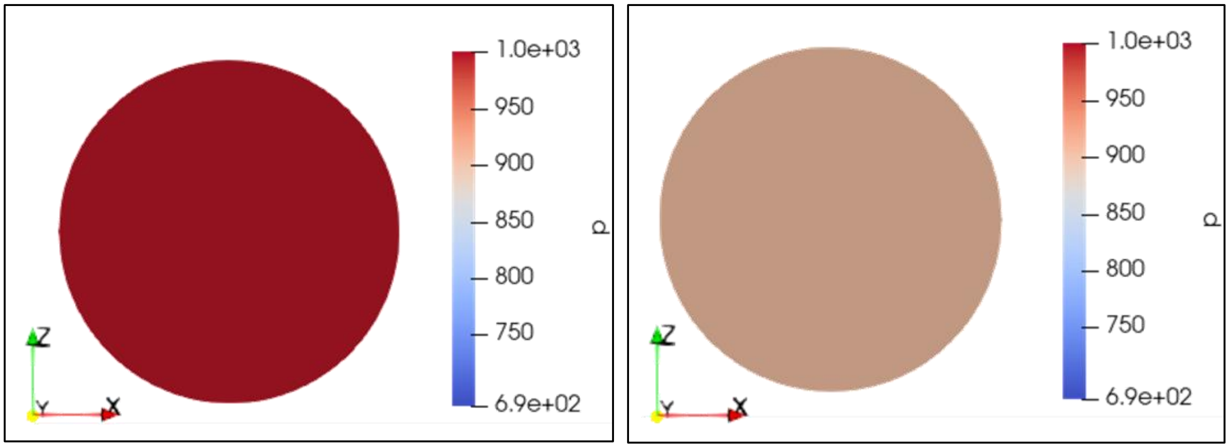

Figure 9: Kinematic pressure contours for inlet and outlet

Static pressure can be calculated by multiplying kinematic pressure by density of water which gives 1000 kPa at inlet and 900 kPa at outlet. Figure 10 and 12 describe pressure and velocity plots at the mid-plane. Maximum resultant velocity of 20 m/s is observed inside baffle section.

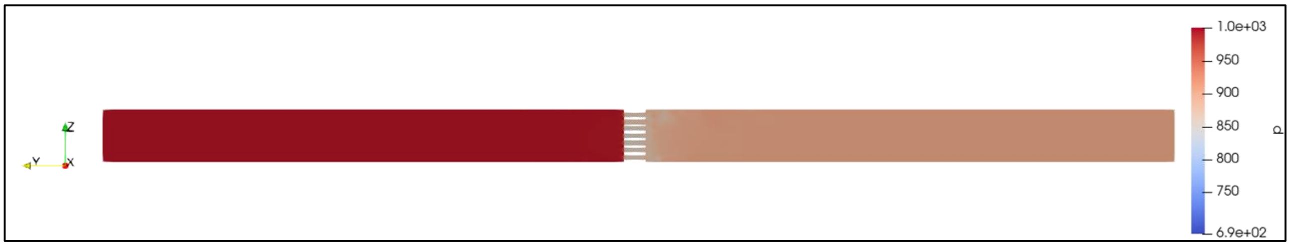

Figure 10: Kinematic pressure plot at mid-plane passing through pipe

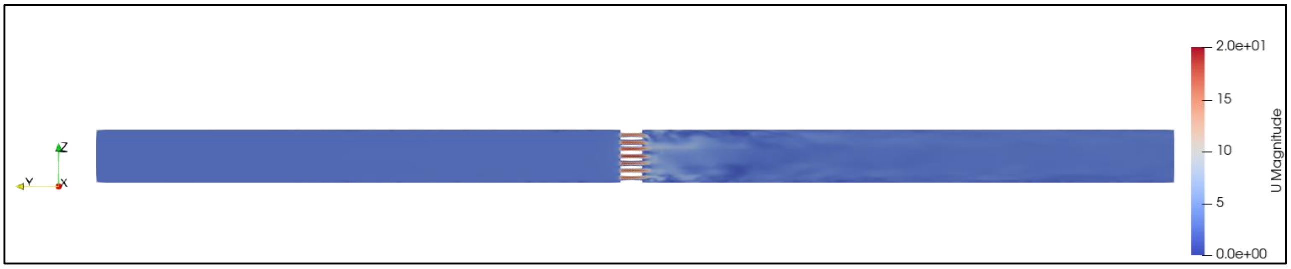

Figure 11: Resultant velocity plot at mid-plane passing through pipe

Calculated value of volume flow rate from CFD analysis is 1.251e-02 and thus, gives mass flow rate of 12.51 kg/s.

Flow coefficient (Cv) is calculated as per the Eq. (4)

Where, Q is flow rate (GPM),

P1 is inlet pressure (psi),

P2 is downstream pressure (psi),

SG is specific gravity.

$$Cv = Q \times \sqrt{\frac{SG}{(P1 - P2)}} \ldots\ldots\ldots\ldots\ldots\ldots Eq. (4)$$

$$Cv = 198.29 \times \sqrt{\frac{1}{(14.5038)}} = \mathbf{52.1}$$

Thus, baffle plate inside pipe gives rated flow coefficient equal to 52.06 as per ISA 75.01.01.

- **Part C:**

The pressure, velocity and temperature distribution at mid-plane passing through the pipe for study part C are presented below. The results show that, pressure drop required to flow 0.01 kg/s of superheated steam at 623°C through the baffle is approximately 0.00287 bar. Maximum velocity of 38 m/s can be observed at the baffle section.

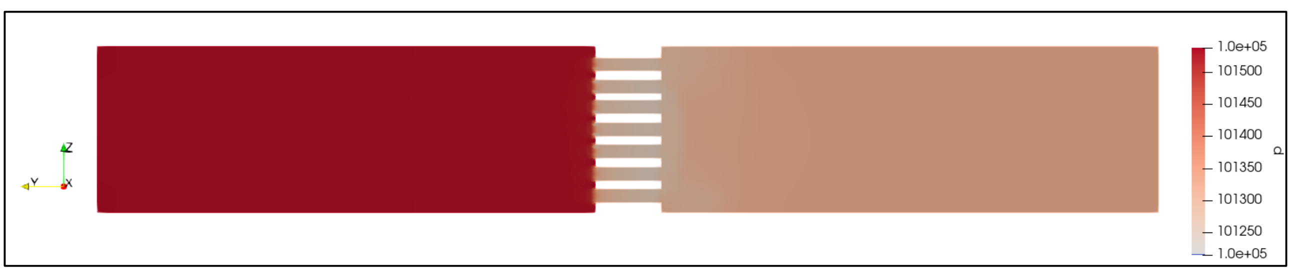

Figure 12: Static pressure plot at mid-plane passing through pipe

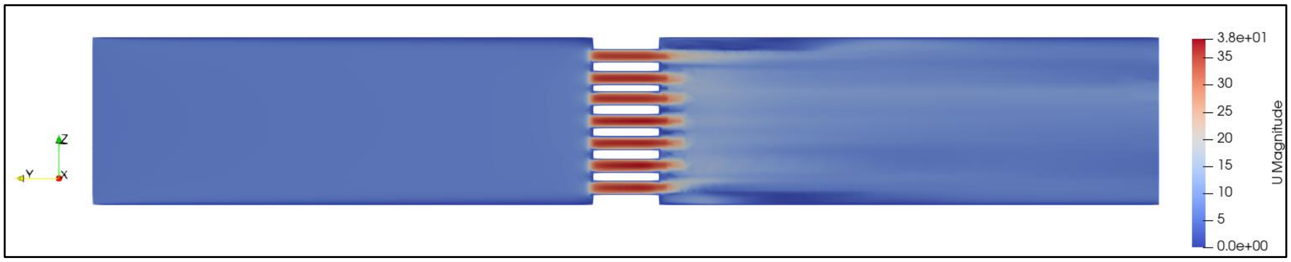

Figure 13: Resultant velocity plot at mid-plane passing through pipe

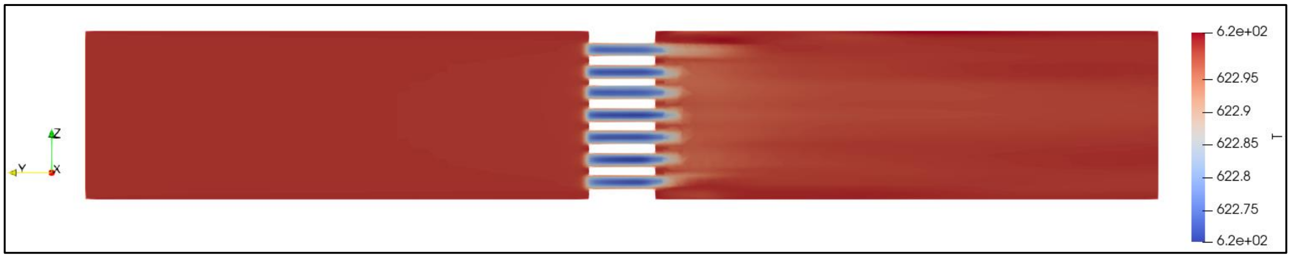

Figure 14: Temperature distribution at mid-plane passing through pipe

Flow coefficient (Cv) is calculated as per the Eq. (5) for non-choked turbulent flow without attached fittings as per ISA-75.01.01 applicable if $x < F_\gamma \, x_T$

$$Cv = \frac{W}{N6Y\sqrt{xP1\rho1}} \dots \dots \dots \dots \dots \dots Eq.(5)$$

Table 5: Flow coefficient (Cv) calculations as per ISA-75.01.01 for part C study

Parameter	**Symbol**	**Value**	**Units**
Mass flow rate at inlet (Computed)	W	36	kg/hr
Density at inlet (Computed)	ρ_1	0.3536	kg/m^3
Absolute Pressure at inlet	P_1	101.537	kPa
Absolute Pressure at outlet	P_2	101.250	kPa
Pressure differential ratio factor	x_T	0.74	-
Numerical Constant	N_6	2.73	-
Specific heat ratio	γ	1.30	-
Specific heat ratio factor	$F\gamma$	0.927	-
Pressure differential ratio	$X = \frac{P1 - P2}{P1}$	0.003	-
Expansion Factor	$Y = 1 - \frac{X}{3F\gamma XT}$	0.999	-
Coefficient of flow From Eq. (5)	C_v	**41.5**	-

From the above calculations, it can be observed that flow coefficient for the process condition using superheated steam is 41.5.

- **Part D:**

This part of the study involves transient analysis carried out till 1 sec of flow time. Initialization of the flow field is done from steady state simulation carried out in part C of the study. The results for transiently varying mass flow rates at inlet and outlet, resultant velocity variation at three probe locations and resultant velocity plots/graphs over the line drawn from inlet to outlet is documented in this study.

Following figure 15 shows the results for mass flow rate variation with respect to time.

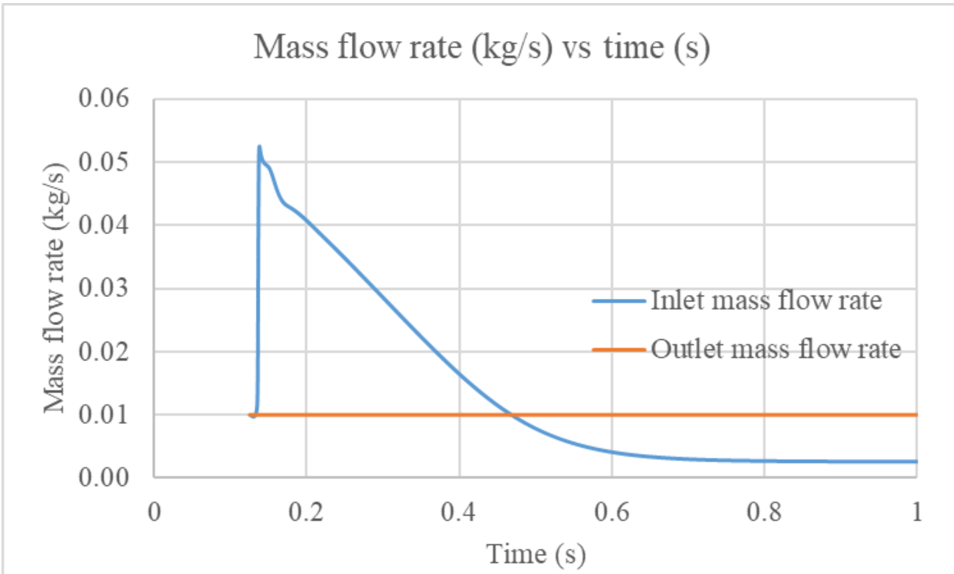

Figure 15: Mass flow rates vs time

Variation in axial velocity at three locations/coordinates mentioned below, is presented in figure 16.

- Probe 0 (0, -0.64, 0) approximately one pipe diameter before baffle
- Probe 1 (0, -0.745, 0) locating inside baffle
- Probe 2 (0, -0.86, 0) approximately one pipe diameter after baffle

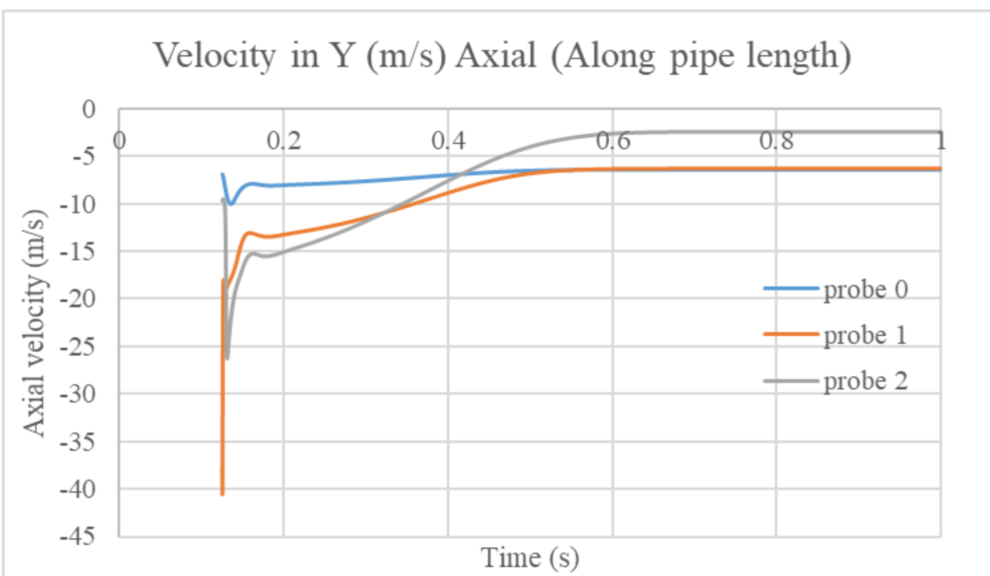

Figure 16: Axial velocity (Y direction) vs time

Following figures show the resultant velocity plotted on the line passing from two points mainly on inlet and outlet, and through the center of the pipe at different time steps.

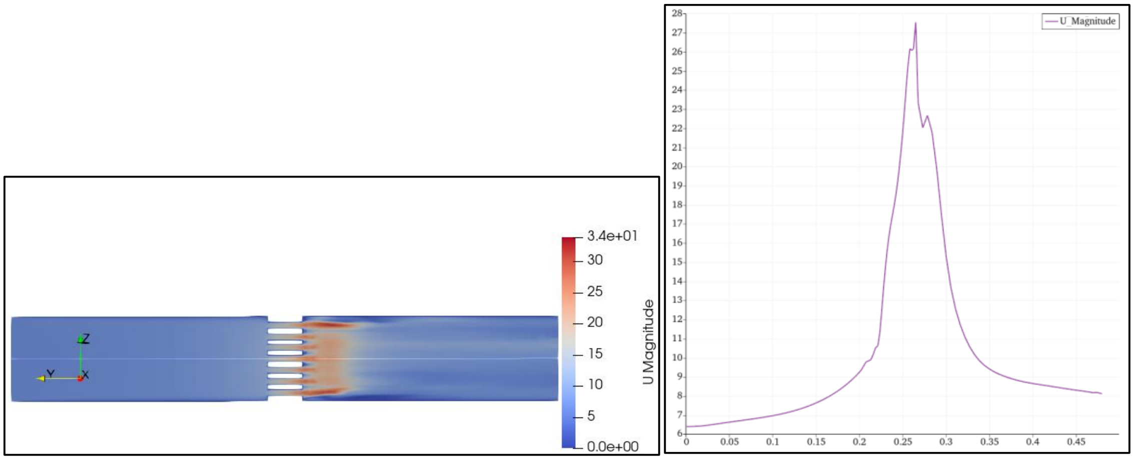

Figure 17: Resultant velocity (m/s) vs time (s) over the line at 0.127 s

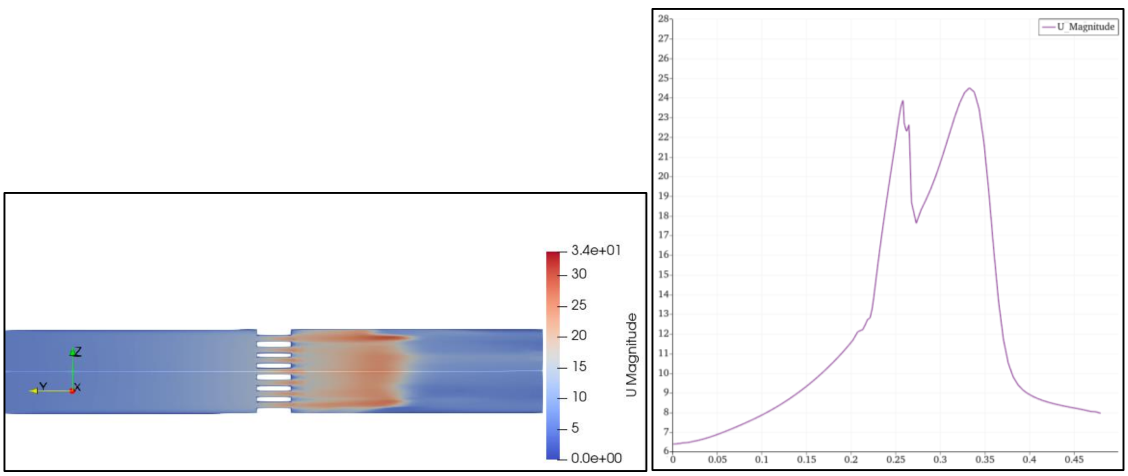

Figure 18: Resultant velocity (m/s) vs time (s) over the line at 0.13 s

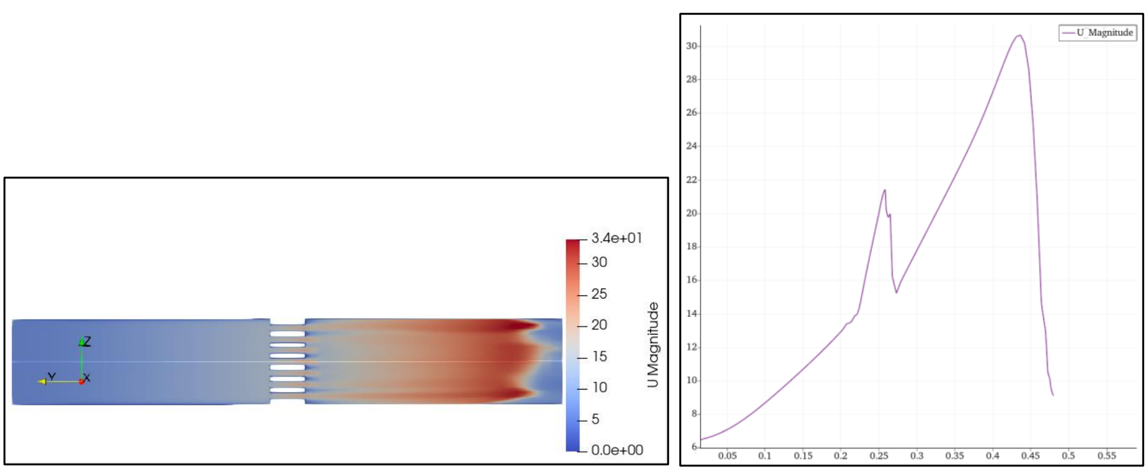

Figure 19: Resultant velocity (m/s) vs time (s) over the line at 0.135 s

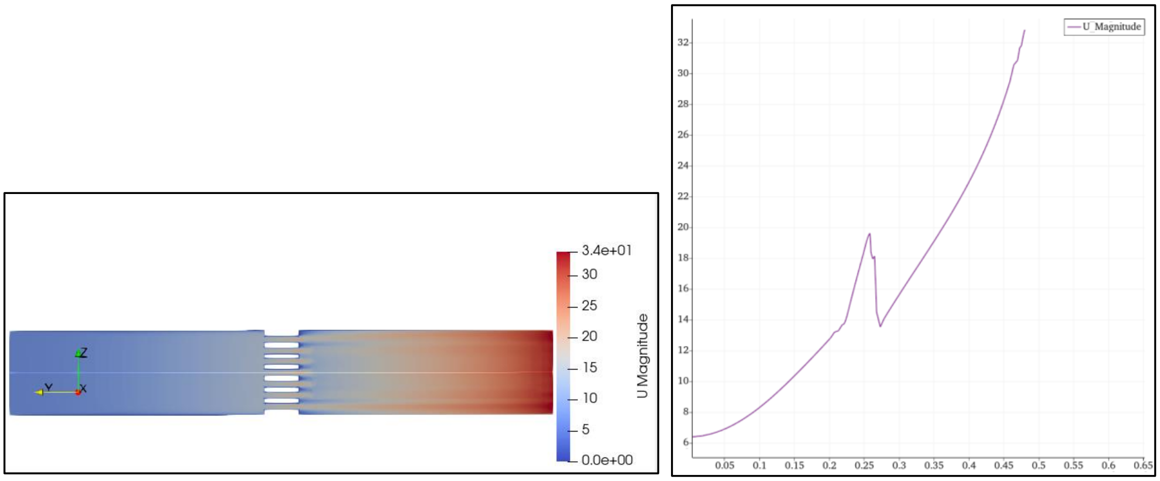

Figure 20: Resultant velocity (m/s) vs time (s) over the line at 0.14 s

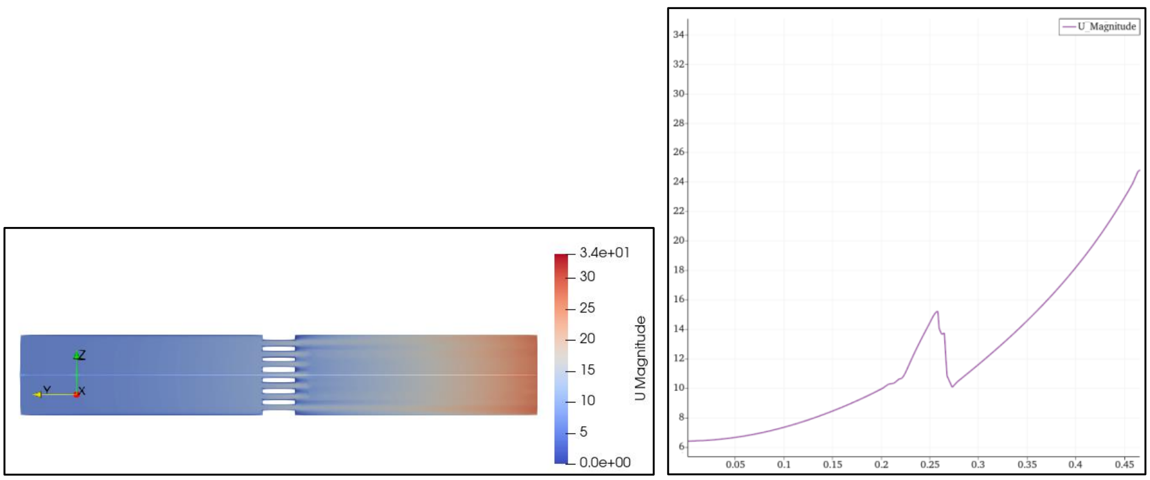

Figure 21: Resultant velocity (m/s) vs time (s) over the line at 0.2 s

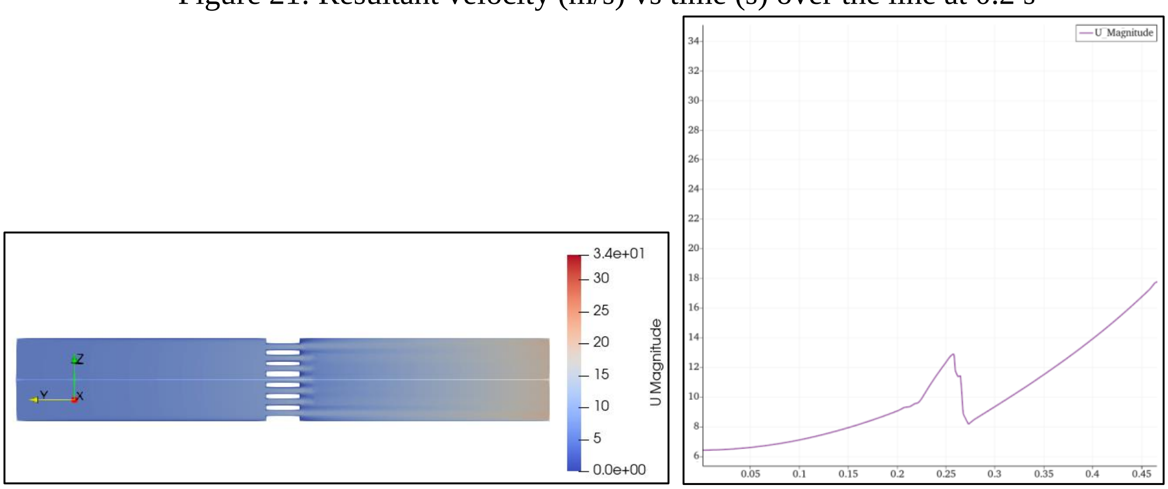

Figure 22: Resultant velocity (m/s) vs time (s) over the line at 0.3 s

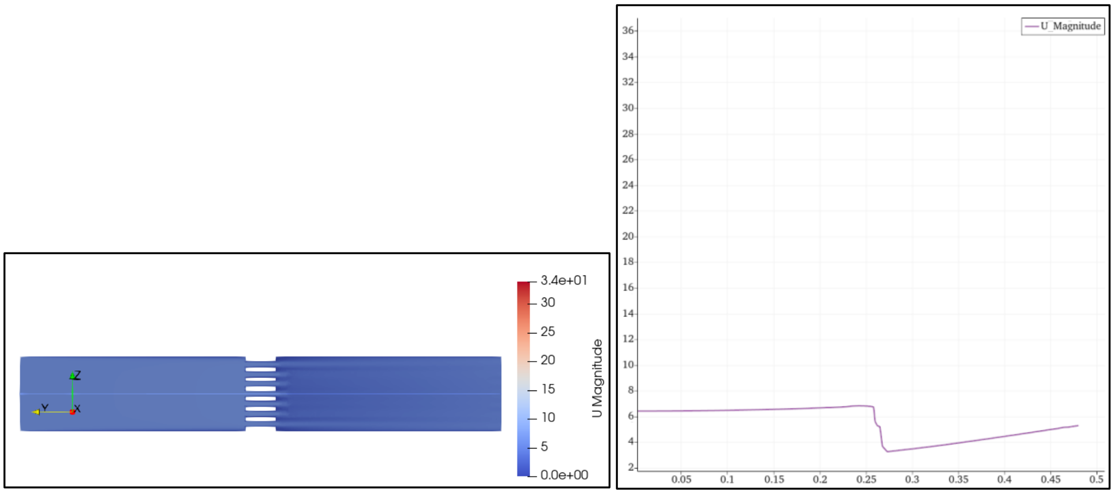

Figure 23: Resultant velocity (m/s) vs time (s) over the line at 0.5 s

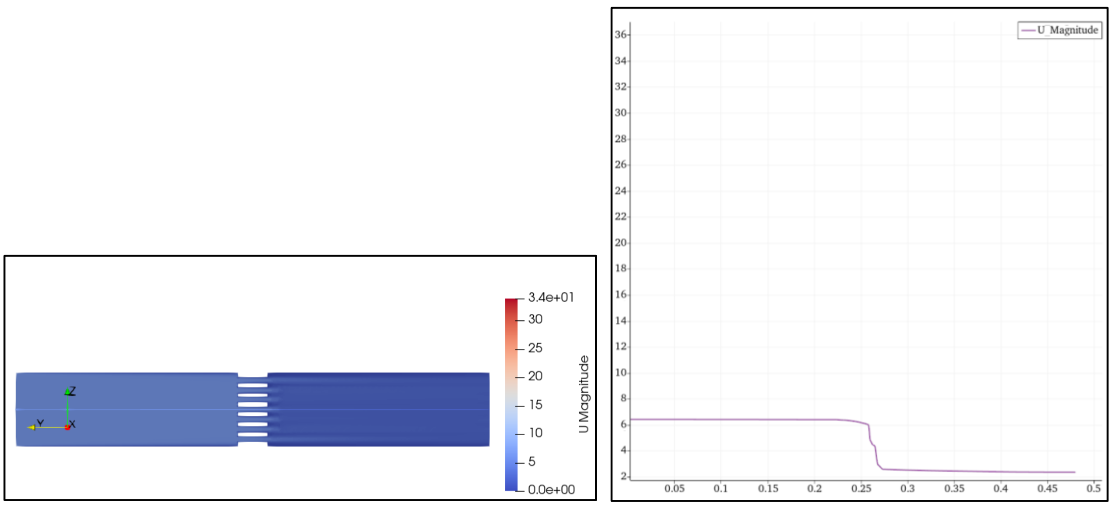

Figure 24: Resultant velocity (m/s) vs time (s) over the line at 0.7 s

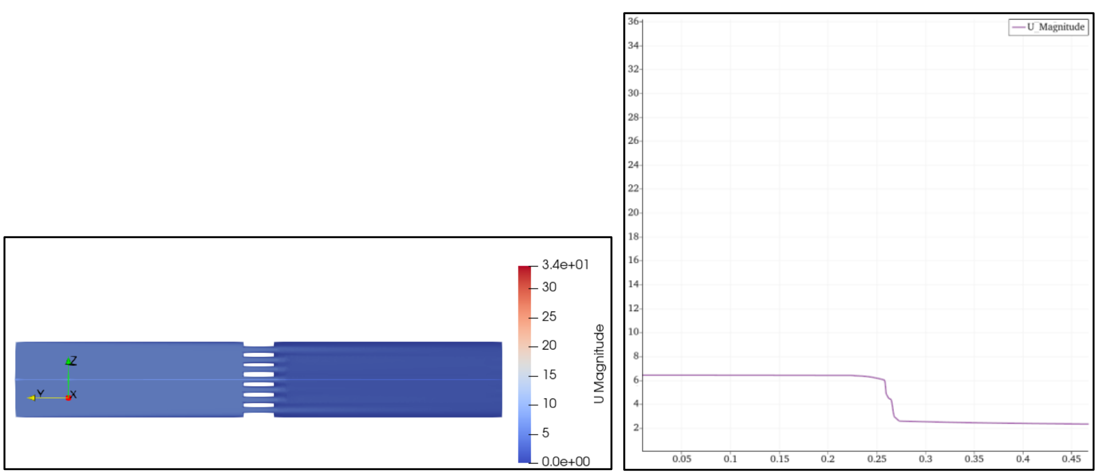

Figure 25: Resultant velocity (m/s) vs time (s) over the line at 0.85 s

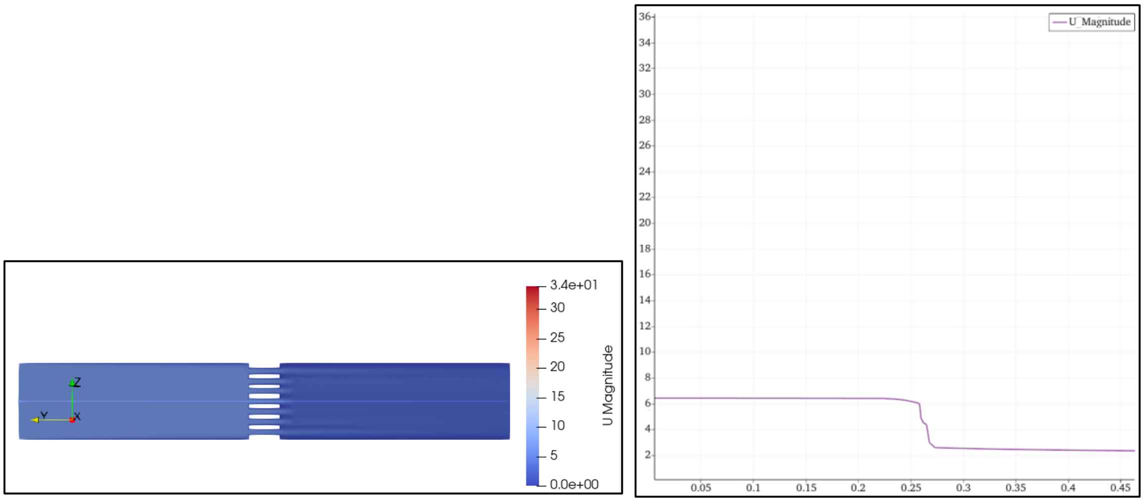

Figure 26: Resultant velocity (m/s) vs time (s) over the line at 1 s

Thus, comparative assessment is successfully carried out between the paper and present simulation in OpenFOAM for three fluids with different viscosities in part A of the study, which presents laminar flow profile for high viscosity fuel-oil, and turbulent flow profile for low viscosity water and alcohol. This study is extended to further where, rated flow coefficient (Cv) is calculated for baffle introduced at center of the pipe for water as fluid in part B. Study Part C includes flow coefficient calculations carried out using steady state analysis with superheated steam for the process condition. Study part D involves transient analysis for the same geometry, and process condition used in the part C to analyze transiently varying flow parameters.

5 References

List all the references used in creating this report. The reference to the paper that has been used to migrate to OpenFOAM V9 in this project should be cited as the first source.

1. Bejena, Baru & Prabhu S, Venkatesa & Gundaboina, Saikiran. (2021). Computational Fluid Dynamics Simulation and Analysis of Fluid Flow in Pipe: Effect of Fluid Viscosity. Journal of Computational and Theoretical Nanoscience. 18. 805-810. 10.1166/jctn.2021.9680.

2. ANSI/ISA–75.01.01–2002 (IEC 60534-2-1 Mod). (2002). Flow Equations for Sizing Control Valves (2002nd ed.).

3. https://doc.cfd.direct/notes/cfd-general-principles/

4. https://www.openfoam.com/documentation/guides/latest/api/classFoam_1_1PengRobinsonGas.html

5. https://www.openfoam.com/documentation/guides/latest/api/classFoam_1_1sutherlandTransport.html

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.