Friction factor in internal pipe turbulent flow

Internship Report
CFD-FOSSEE Team
Indian Institute of Technology, Bombay

Prepared by
Tinto Thomas
TKM College of Engineering, Kollam

Under the supervision of

Prof. Manaswita Bose

INDIAN INSTITUTE OF TECHNOLOGY, BOMBAY

ACKNOWLEDGEMENT

The following report was created as a part of the FOSSEE semester-long internship and I would like to thank FOSSEE, Indian Institute of Technology, Bombay for giving me this opportunity.

I would like to thank my project guide Prof. Manaswita Bose, and mentor Mr. Ashuthosh P Shridhar for the support in carrying out simulations throughout the internship. I would also like to thank the project manager, Ms. Payel Mukherjee, for giving me this opportunity.

Tinto Thomas
TKM College of Engineering, Kollam
Date: June 18, 2022

Contents

1 Introduction **1**

1.1 Aim 1

1.2 Objective 1

1.3 Theory 2

1.4 Literature Review 6

2 OpenFOAM base case **7**

2.1 Base Case structure 7

2.1.1 Laminar model 7

2.1.2 Turbulent model 8

2.2 Solver – simpleFoam 9

3 OpenFOAM Case Modifications **10**

3.1 Wedge Geometry and Mesh 10

3.2 Programmable blockMeshDict 11

3.3 Smooth Pipe 13

3.3.1 Laminar flow 13

3.3.1.1 Pre-processing & Boundary Conditions 13

3.3.1.2 Control parameters, fvSchemes & fvSolution 16

3.3.1.3 Post-processing 17

3.3.2 Turbulent flow 19

3.3.2.1 Pre-processing & Boundary Conditions 19

3.3.2.2 Control parameters, fvSchemes & fvSolution 22

3.3.2.3 Post-processing 25

3.4 Rough Pipe 26

3.4.1 Turbulent flow 26

3.4.1.1 Pre-processing & Boundary Conditions 27

3.4.1.2 Control parameters, fvSchemes & fvSolution 28

3.4.1.3 Post-processing 29

4 Results **30**

4.1 Smooth Pipe 30

4.1.1 Laminar flow 30

4.1.1.1 Analytical Results 30

4.1.1.2 Numerical Results 31

4.1.1.3 Result validation 31

4.1.2 Turbulent flow 33

4.1.2.1 Empirical Results 33

4.1.2.2 Numerical Results 34

4.1.2.3 Result validation /comparison 35

4.2 Rough Pipe 38

4.2.1 Turbulent flow 38

4.2.1.1 Analytical / Empirical Results 38

4.2.1.2 Numerical Results 39

4.2.1.3 Result validation / comparison 40

5 Observation **43**

6 Conclusion **45**

7 References **47**

Chapter 1

Introduction

1.1 Aim

The lab migration project aims to study the frictional losses for varying Reynold's numbers in smooth and rough straight pipe internal flows for laminar and turbulent flows. Results from OpenFOAM simulations will be validated against analytical / empirical results.

1.2 Objective

Wedge geometry for the cylindrical pipe was made using coded blockMeshDict in OpenFOAM. Entire simulations were done in Openfoamv7 and post-processed using Paraview 5.6.0. GNU nano and Gnuplot were used for editing the files and plotting graphs. At a particular stage of simulation, what-if analysis in MS Excel was used to find iterative solutions. MS Excel was also used for calculations and plotting certain graphs.

Frictional losses in the smooth and rough straight pipe internal flows are meant to be studied in this project. We will use water as the fluid for analysis and hence will adopt the physical property values of the same. Instead of a 2D or 3D pipe geometry, we will be using a wedge geometry. The geometry with a small wedge angle will be created using the programmable blockMeshDict utility. Programmable blockMeshDict will favor us to create different dimension pipes very easily. However, in this project, we will keep a geometry of fixed dimension throughout the simulations. Reynold's number will be varied for different cases through varied inlet velocity. Respective analytical / empirical velocity profiles will be applied for both laminar and turbulent flows at the pipe inlet as a boundary condition. SimpleFOAM is the OpenFOAM solver we will be using which is an incompressible, steady-state solver that can be used for analyzing both laminar and turbulent flows. Navier-Stokes equations will be used for the laminar model while the k-omega SST RANS model will be used for the turbulent model. However, we will start with the k-epsilon model and thus it can be used to understand the performance of the k-epsilon and k-omega SST models. There is also a provision available for us to move into different divergent schemes used in fvSchemes. By applying different divSchemes, the changes in results can be noted and analyzed. The result could be utilized in further studies. The analysis study is carried out separately for smooth and rough pipes. This was done by using nutKWallFunction and nutKRoughWallFunction for smooth and rough pipes respectively. According to the results, roughness height parameter Ks can be changed and the trend can be recorded. This data can be used to understand the working of nutKRoughWallFuntion in rough pipes. Velocity profile at any point and time can be plotted and the resultant profile at the outlet can be used as a validation method to check the flowing nature. Velocity profile check, Maximum velocity value, pressure drop calculations, contours, and plot patterns can be obtained from the results. Validations can be done against analytical, experimental, and empirical results like Reynold's equation, Darcy-Weisbach equation, Moody chart, power-law profile, Colebrook equation, and so on.

1.3 Theory

Laminar flows are smooth and streamlined, whereas turbulent flows are irregular and chaotic. The flow behavior drastically changes from laminar to turbulent flow. There comes a transition state in between these two which is very complex to predict accurately. Reynolds number is an important dimensionless parameter that identifies the behavior of fluid based on attributes like viscosity or velocity of the fluid. Accordingly, the value of Reynolds number (Re) can be expressed as:

$$\mathrm{Re} = {}^{\rho u D}\!/_{\mu}$$

where ρ = density of fluid ; u = velocity of flow ; D = diameter of pipe ; μ = dynamic viscosity of the fluid

Inlet flow velocity is one of the major factors which decides the nature of the flow. Accordingly, it takes a particular velocity profile during the flow. If the flow is laminar, the velocity distribution at a cross-section will be parabolic in shape with the maximum velocity at the center being about twice the average velocity in the pipe. The velocity profile in turbulent flow is flatter in the central part of the pipe than in laminar flow. The flow velocity drops rapidly, extremely close to the walls. This is due to the diffusivity of the turbulent flow. These profiles can be predicted to an extent utilizing analytical / empirical equations.

For laminar flow,

$$u = 2 \text{ x } u_{avg} \, [1 - (\frac{r^2}{R^2})]$$

For turbulent flow, according to Nikuradse's empirical power law equation,

$$u = u_{max} \, [(1 - \frac{r}{R})^{1/n}]$$

$1/n = 0.338 \, Re^{-0.081}$, however, for low Reynold's number flows, 1/n = 1/6

where, u_{max} = maximum velocity ; u_{avg} = average velocity ; u = velocity at any point ; r = distance from centre ; R = pipe radius.

Relation between maximum velocity and average velocity for laminar and turbulent flow are given.

$$u_{max} = 2 \text{ x } u_{avg} \text{ for laminar flow and } u_{max} = \frac{u_{avg}}{0.8} \text{ for turbulent flow}$$

We can apply this velocity profile at the inlet as a boundary condition. This will help our simulation to attain steady state easily and thus get converged with a lesser number of iterations.

A rough pipe is different from a smooth pipe such that a specific level of surface roughness is present on the walls of the pipe. This roughness can be of different profiles, average height, and distribution. According to these parameters, flow velocity, pressure drop, shear velocity, friction factor, and so on could be different from one to another. In this project, a straight pipe of stainless steel will be considered for analysis of frictional losses. For rough pipe analysis, an average roughness height of 0.015mm is taken, considering stainless steel pipe material. The roughness profile is assumed to be sand grain roughness with uniform distribution throughout the surface of the pipe.

For turbulent flow in smooth pipes, the nutkWallFunction boundary condition provides a wall constraint on the turbulent viscosity, i.e., nut, based on the turbulent kinetic energy. The nutkWallFunction condition inherits the traits of the nutWallFunction boundary condition. When it comes to rough pipes, nutKRoughWallFunction will be used as the boundary condition instead of nutKWallFunction. The condition manipulates the wall roughness parameter, to account for roughness effects. The nutkRoughWallFunction condition inherits the traits of the nutkWallFunction boundary condition. Two parameters that should be mentioned are roughness height, Ks, and roughness constant, Cs. Ks account for the average surface roughness height whereas Cs stand for the roughness distribution along the pipe walls. Ks value, 0 indicates smooth pipe and the value varies according to the pipe surface roughness height. Cs value lies between 0.5 – 1, where 0.5 is the uniform roughness distribution and the value varies according to the uneven roughness distribution.

The friction factor represents the loss of pressure of a fluid in a pipe due to the interactions between the fluid and the pipe. The loss in energy of fluids while traveling through a pipe or ducts causes a reduction in pressure and velocity which is known as head loss. There are three main factors affecting friction – pipe diameter, Reynold's number, and surface roughness. In this study, we keep pipe diameter and surface roughness constant while Reynold's number varied by changing inlet fluid velocity. Reynold's number for the flow depends on the flow velocity, fluid density and viscosity, and pipe length.

The Darcy-Weisbach equation is used to estimate the head loss, h_f for a fluid flowing at a velocity u, in a pipe having length l, diameter d, and friction factor f, such that :

$$h_f = \frac{f l {u_{avg}}^2}{2gd}$$

For laminar flow in smooth pipes, from the analytical equation, the friction factor can be calculated using,

$$f = \frac{64}{Re}$$

When it comes to the turbulent flow, the empirical relation used for calculating the friction factor for smooth pipes is,

$$f = \frac{0.316}{Re^{0.25}}$$

The Colebrook-White equation is used as an empirical formula to calculate the friction factor for turbulent flow in rough pipes. It can only be solved using numerical approximations. The iterative method should be adopted to find an approximate solution for the friction factor.

$$\frac{1}{\sqrt{f}} = -2\log_{10}\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right)$$

where, f = friction factor ; Re = Reynold's number ; ε = roughness height ; D = diameter of pipe ; $\frac{\varepsilon}{D}$ = relative roughness

To find the friction factor from a Moody chart, you need values for Reynold's number and the relative roughness (k/D). Trace the relative roughness curve and draw a line from Reynold's number on the x-axis. The point where Reynold's number line intersects the roughness curve gives the Moody friction factor. The relative roughness of a surface can be calculated using the formula, k/D, where k is the surface roughness and D is the hydraulic diameter.

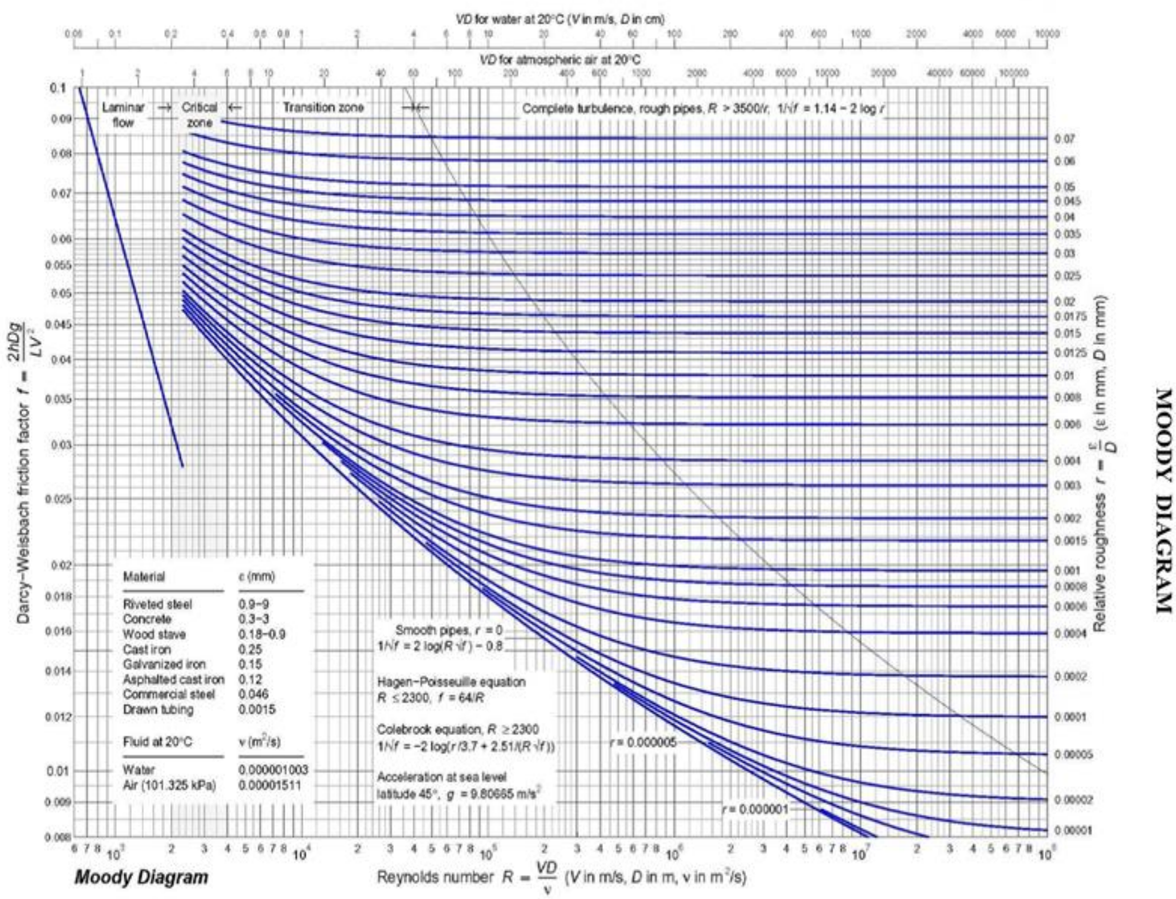

Fig 1.3.1 Moody Chart

1.4 Literature Review

One of the most reviewed literatures on frictional losses in straight pipe internal flow was *Numerical Analysis of friction factor for a fully developed turbulent flow using k-epsilon turbulence model with enhanced wall treatment - Muhammad Ahsan (2014).* This paper conducted a study on frictional losses which is almost similar to our problem description. The geometry setup used in the reference paper was 20m in length and 1m in diameter. An analysis study was conducted for the steady-state incompressible flow of water in the range of Re10000 – Re40000 with the k-epsilon RANS model. The observations and results from the study are following. The meshes with cell sizes 1mm, 2mm, and 3mm didn't create much difference in the results since the geometry is simple. Fluid flow is not that complex which facilitates the use of higher order discretization schemes if needed. The paper demanded fine working of the k-epsilon model with enhanced wall treatment in addition to the good agreement of computed values with experimental values while validation.

Chapter 2

OpenFOAM base case

2.1 Base Case structure

2.1.1 Laminar model

Laminar flow is characterized by fluid particles following smooth paths in layers, with each layer moving smoothly past the adjacent layers with little or no mixing. With the help of analytical results, perfect solutions for laminar flow are possible. Out of five simulations, two of them (Re1000 and Re2500) were analyzed under laminar modeling. Re1000 flow is a case of laminar flow when Re2500 falls under the category of transitional flow. Even though, we must consider the fact that Re2500 flow just entered the transitional stage. Hence, there is a high chance that the flow still delivers all the characteristics of laminar flow and behaves like one. All the simulation runs were performed using OpenFOAM. A base case that exhibits good similarity with our problem statement was chosen from the tutorials folder in OpenFOAM. Our flow is considered to be laminar, incompressible, and steady state. Hence tutorials/ incompressible/ simpleFoam/ pipecyclic was picked.

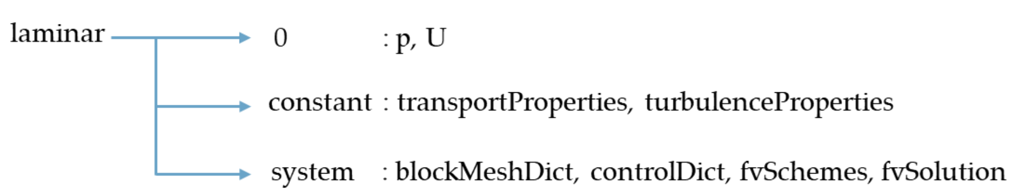

Fig 2.1.1.1 OpenFOAM case Structure - Laminar

Laminar flow is independent of the pipe roughness since the flow is stratified and covers the roughness. It then behaves like a flow along a smooth wall. Hence, rough pipe cases will not be analyzed with laminar flow models in our study.

2.1.2 Turbulent model

Turbulent fluid motion is an irregular condition of the flow in which quantities show a random variation with time and space coordinates so that statistically distinct average values can be discerned. Hence, expecting an ideal result to the real case scenario is illogical as of now. Different approaches must be used to find a close match with the empirical / experimental results. It was preferred to use the k-omega SST RANS model for our turbulent flow models. The decision was made after reviewing results from similar analysis studies. However, it is open to check the result's accuracy using any other RANS models say, k-epsilon in our case. Turbulent model samples from our analysis study, Re5000 and Re10000 flows were simulated using the k-omega SST model. Re3500, which is a transitional flow is also checked with the same setup. The flow is considered to be turbulent, incompressible, and steady state. Hence, a close match base case was found from OpenFOAM, tutorials/ incompressible/ simpleFoam/ pitzDaily. Epsilon directory was picked up for the k-epsilon model of turbulent flow, which was later replaced by the omega directory for the k-omega SST model.

K-epsilon RANS
- → 0 : p, U, k, epsilon, nut
- → constant : transportProperties, turbulenceProperties
- → system : blockMeshDict, controlDict, fvSchemes, fvSolution

Fig 2.1.2.1 OpenFOAM case structure – k-epsilon RANS

K-omega SST RANS
- → 0 : p, U, k, omega, nut
- → constant : transportProperties, turbulenceProperties
- → system : blockMeshDict, controlDict, fvSchemes, fvSolution

Fig 2.1.2.2 OpenFOAM case structure – k-omega SST RANS

Re3500 falls under the transitional flow regime very clearly. The flow must exhibit a mixture property of laminar and turbulent flows. There is no actual method to find and validate the results of these flows. Even though it is not yet a complete turbulent flow, we could say that it must be very near to that. Hence, here we adopted an approach to check how close is Re3500 to the turbulent flow regime. This case is analyzed with a turbulent model setup and validated against the same. Percentage error could say how close is the transitional flow to the turbulent flow.

The situation is different from laminar flow if the flow gets turbulent. A very small roughness could be covered completely by the laminar sublayer. Hence there is a significance for turbulent flows in the rough pipes. We will analyze turbulent flow models of Re5000 and Re10000 with the rough pipe. From analyzing the results, the study can be extended by focusing on the nutKRoughWallFunction directory which replaced the nutKWallFunction directory of smooth pipe turbulent flow cases.

2.2 Solver - simpleFoam

SimpleFoam assumes an incompressible, steady state, viscous flow and can be applied for both laminar and turbulent flow modeling. It uses the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm.

The solver employs the **SIMPLE** algorithm to solve the continuity equation:

$$\nabla \cdot \mathbf{u} = 0$$

and momentum equation:

$$\nabla \cdot (\mathbf{u} \otimes \mathbf{u}) - \nabla \cdot \mathbf{R} = -\nabla p + \mathbf{S}_u$$

Where:

$\mathbf{u}$ = Velocity
p = Kinematic pressure
$\mathbf{R}$ = Stress tensor
$\mathbf{S}_u$ = Momentum source

Fig 2.2.1 Governing equations (OpenFOAM User Guide)

Chapter 3

OpenFOAM Case Modifications

The entire simulation study is done by making modifications to the directories and dictionaries of a base case folder which is taken from the OpenFOAM tutorials which closely resembles a model of our kind. After picking up a base case set up for the simulation, necessary changes are made to the dictionaries. Certain functions are added and parameters are altered according to our flow problems. One of the greatest advantages of using OpenFOAM is that the approach and methodology to the results is a free choice for the user. We are free to make assumptions and can simplify our flow problem domain in such a way that all physical and mechanical properties are directly involved or substituted. Anything that affects our solution cannot be simply eliminated.

3.1 Wedge Geometry and Mesh

Geometry and mesh play a crucial role in every analysis study. No matter how well we capture the geometry, the poor mesh will always result in inaccurate and improper solutions. Similarly, if we couldn't capture the geometry accurately, no fine mesh could provide better results. We have used wedge geometry to analyze the pressure drop and frictional losses in the smooth and rough straight pipe internal flows.

Wedge geometry is simple like 2D geometry, yet behaves like 3D and gives better results than 2D. One big advantage of wedge is that it allows for the rotational flow around the axis and this is possible only because the fluxes and normal components are not assumed as zero. Wedge is different from the symmetry boundary condition in the sense that the wedge BC applies the Navier-Stokes equations in cylindrical coordinates whereas the latter applies in cartesian coordinates. A very small wedge angle will be used for creating geometry. For our flow problem to be analyzed, we needed a cylinder with 0.00344m diameter and 0.1m length. We have considered a wedge geometry with wedge angle of 4degrees to substitute our straight pipe and made it with the help of the programmable blockMeshDict utility. Proper meshing is also done by using the same utility.

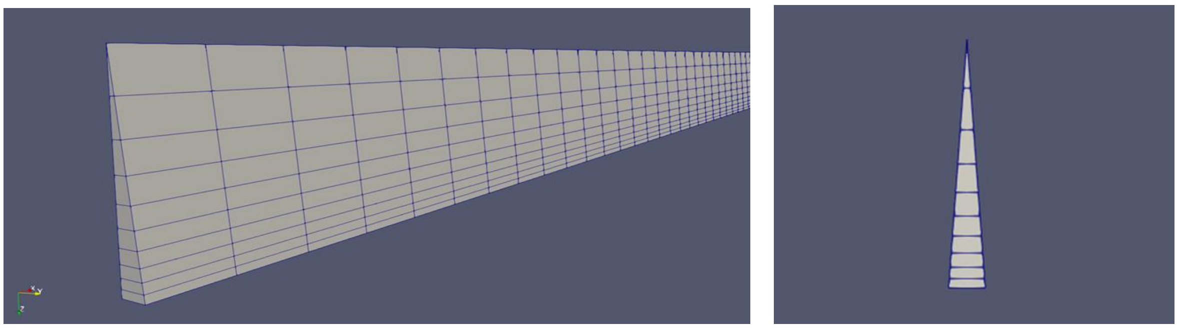

Fig 3.1.1 Wedge geometry and Mesh

3.2 Programmable blockMeshDict

Programmable blockMeshDict is something that can make the geometry easily, using the program codes by changing the domain's physical dimensions only. This is a modification to the normal blockMeshDict utility, in which instructions are given using codes now. We define the origin points initially, followed by the domain radius, length, and wedge angle. These are the parameters that can be easily modified by giving the values directly. Hence many more pipe domains of different dimensions can be easily generated by altering these three values from programmable blockMeshDict. This facilitates the analysis study to be extended to different pipes very easily. The complex coordinates needed to define the vertex and edges can be easily calculated using this utility. The length and radius of the pipe are defined along the X axis and Z axis respectively. Only one cell is placed along the Y axis which is the wedge side. Hence OpenFOAM doesn't solve equations in that axis, just like the 2D analysis setup. The number of elements along different axis can be defined within the utility. The mesh is done by placing 100 cells along the length of the pipe and 20 cells along the radius. The space between cells can be adjusted using simpleGrading. Cells along the length are kept undisturbed and given equal spacing. Cells along the Z axis are modified so that cells at the wall will be denser than the cells at the center of the pipe. This helps to introduce wallfunctions, which could provide good results at the walls of the pipe. After the generation of mesh, the quality is checked using the checkMesh command, which gave very satisfactory data about grids. Thus, a fine quality mesh is ensured for the simulations. Since the quality of the mesh plays a great role in delivering accurate results, checkMesh command is very useful at times.

```
convertToMeters 1;

cx 0; //origin x
cy 0; //origin y
cz 0; //origin z

r 0.00172; //radius of pipe (dia 3.44mm)
l 0.1; //length of pipe
a 4; //angle in degree

rada   #calc "degToRad($a)";
y1     #calc "$r*sin($rada)";
y2     #calc "-1*$y1";
z1     #calc "$r*cos($rada)";


vertices
(
    ($cx $cy $cz)
    ($l $cy $cz)
    ($cx $y2 $z1)
    ($l $y2 $z1)
    ($l $y1 $z1)
    ($cx $y1 $z1)
);

blocks
(
    hex (0 1 1 0 2 3 4 5) (100 1 20) simpleGrading (1 1 0.2)
);

edges
(
    arc 2 5 ($cx $cy $r)
    arc 3 4 ($l $cy $r)
);
```

Fig 3.2.1 Coded blockMeshDict

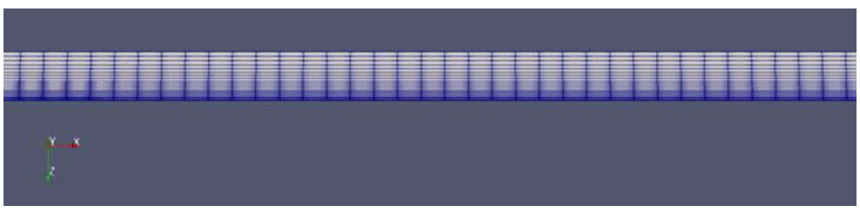

Fig 3.2.2 Wedge geometry - Mesh

```
Overall number of cells of each type:
    hexahedra:     1900
    prisms:        100
    wedges:        0
    pyramids:      0
    tet wedges:    0
    tetrahedra:    0
    polyhedra:     0

Checking topology...
    Boundary definition OK.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
    Number of regions: 1 (OK).

Checking patch topology for multiply connected surfaces...
    Patch               Faces    Points   Surface topology
    inlet               20       41       ok (non-closed singly connected)
    outlet              20       41       ok (non-closed singly connected)
    top                 0        0        ok (empty)
    front               2000     2121     ok (non-closed singly connected)
    back                2000     2121     ok (non-closed singly connected)
    bottom              100      202      ok (non-closed singly connected)

Checking geometry...
    Overall domain bounding box (0 -0.000119981 0) (0.1 0.000119981 0.00171581)
    Mesh has 2 geometric (non-empty/wedge) directions (1 0 1)
    Mesh has 3 solution (non-empty) directions (1 1 1)
    Wedge front with angle 4 degrees
    Wedge back with angle 4 degrees
    All edges aligned with or perpendicular to non-empty directions.
    Boundary openness (7.8585e-21 1.03948e-16 4.2276e-16) OK.
    Max cell openness = 1.56645e-16 OK.
    Max aspect ratio = 29.5806 OK.
    Minimum face area = 2.03824e-09. Maximum face area = 2.39962e-07.  Face area magnitudes OK.
    Min volume = 2.03824e-12. Max volume = 1.30704e-11.  Total volume = 2.05865e-08.  Cell volumes OK.
    Mesh non-orthogonality Max: 0 average: 0
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 0.326845 OK.
    Coupled point location match (average 0) OK.

Mesh OK.
```

Fig 3.2.3 Mesh quality - checkMesh result

3.3 Smooth Pipe

Flow can be either laminar or turbulent in smooth pipes. The flow resistance will be much smaller here than in the rough one. The roughness height of the pipe is assumed to be zero and not specifically mentioned anywhere.

3.3.1 Laminar flow

Flows of Reynold's number 1000 and 2500 are considered here in laminar flow. Re1000 flow is a clear example of laminar flow. Even though Re2500 falls under the transitional flow regime, it is analyzed against the laminar model. It is done by assuming that Re2500 flow is too soon to lose its laminar characteristics. This can also be used as a study topic and extended further.

3.3.1.1 Pre-processing & Boundary conditions

Wedge geometry and mesh are created using programmable blockMeshDict. The number of cells along different axes and spacing can be controlled. Simulations are an approximation of actual results using discretization schemes and numerical methods. As the quantity and quality of the cells increases, the result gets better. But considering computational time and improvement in results, increasing cell count beyond a limit is not encouraged. This is called grid independent study.

Same geometry with 500 cells (100x5), 2000 cells (200x10), and 8000 cells (400x20) were analyzed under the same constraints, and results were studied. 2000 cell was found to be a better deal considering the grid independence study. 8000 cell geometry gave better results but was not worth the computational time considering the improvement of the result from that of the 2000 cell geometry. That is how 2000 cell geometry was finalized for the project work. This trend was followed by both Re1000 and Re2500 flow. For Re1000 flow, maximum velocity value, Umax was found to be 1.0292m/s, 1.04435m/s and 1.04906m/s for 500, 2000 and 8000 cell geometries respectively. A graph was plotted against these values. A steep rise in Umax value can be observed from 500 cells to 2000 cells. But from 2000 cells to 8000 cells, results didn't improve much comparatively.

These data in tabular form and plotted graph are added here for better understanding.

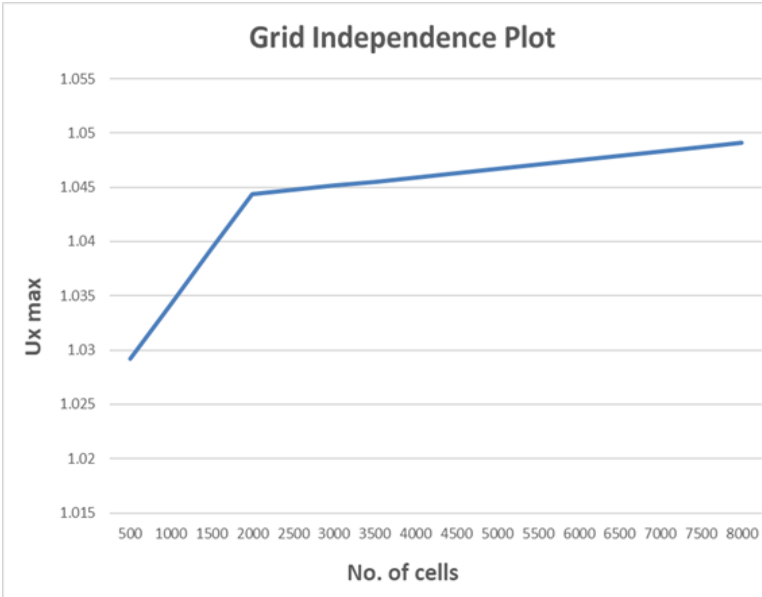

Fig 3.3.1.1.1 Grid independence plot

No. of cells	**Umax (m/s)**
500 (50x10)	1.0292
2000 (100x20)	1.04435
8000 (200x40)	1.04906

Table 3.3.1.1.1 Grid independence table

The boundary conditions used here are mostly similar to any other laminar flow. Wedge BC is applied at both sides of the geometry in both U and p directories. This would help OpenFOAM to calculate results such that for a 3D pipe geometry. The top face (central axis) of the pipe is given symmetryPlane BC everywhere, which could assume the same field opposite the wedge gives better results. To attain steady state fluid flow and results convergence within a lesser number of iterations, a parabolic profile velocity inlet is applied as a boundary condition. This is because laminar flow takes a parabolic velocity profile in its complete flow. The maximum inlet velocity applied is 1.0465m, which is double the average velocity for the Re1000 flow. Similarly, the same BC is applied for Re2500, in which the maximum velocity is calculated as 2.616m. The average velocity of the flow was calculated from Reynold's equation. Maximum velocity was found by taking double the value of average velocity using the analytical equation. The codedFixedValue for the inlet velocity condition is added here.

```
inlet
{
    type            codedFixedValue;
            value               uniform (0 0 0);

            redirectType    parabolicVelocity;
            code
            #{
                    const fvPatch& boundaryPatch = patch();
                    const vectorField& Cf = boundaryPatch.Cf();
                    vectorField& field = *this;

                    const scalar R = 0.00172;
                    const scalar Umax = 1.0465;

                    forAll(Cf, faceI)
                    {
                            const scalar r = Cf[faceI].z();

                            field[faceI] = vector(Umax*(1-pow(r/R,2)), 0 , 0);
                    }
            #};
}
```

Fig 3.3.1.1.2 Coded inlet velocity profile – Re1000

The resultant velocity profile of the codedFixedValue inlet is plotted and given below:

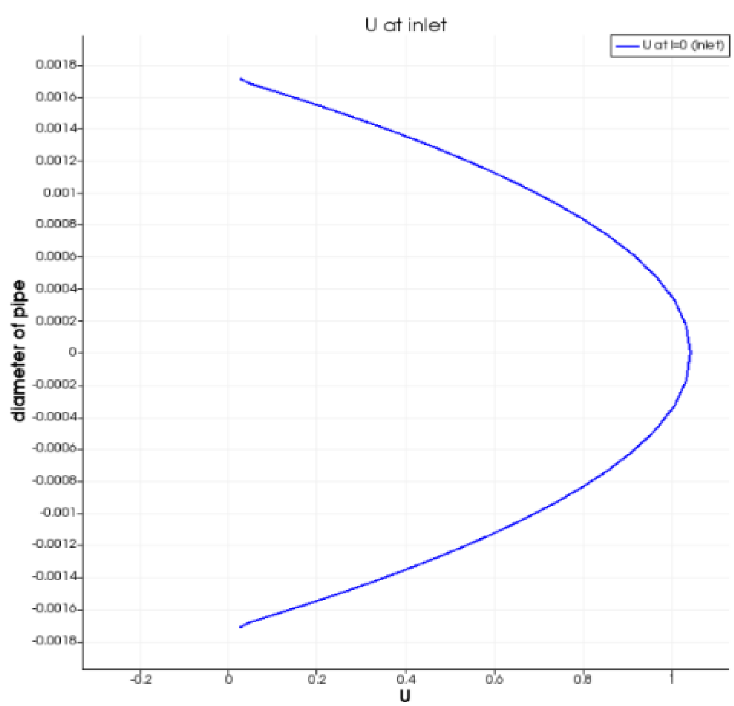

Fig 3.3.1.1.3 Velocity profile at inlet – Re1000

3.3.1.2 Control parameters, fvSchemes & fvSolution

The residual is one of the most fundamental measures of an iterative solution's convergence, as it directly quantifies the error in the solution of the system of equations. Residual measures the local imbalance of a conserved variable. In an iterative numerical solution, the residual will never be exactly zero. However, the lower the residual value is, the more numerically accurate the solution. We can control the residual points for the solution convergence by editing the fvSolution directory. When the residual value hit the pre-mentioned value in fvSolution during the iterations, the solution is said to be converged. Convergence and residual values are considered as major indications of how good is the simulation results. By residuals plot, it is very easy to keep the track of residual values at each iteration and the total number of iterations required for the convergence. The number of iterations required will be different when we change parameters and functions from controlDict, fvSchemes, and fvSolution.

A second sub-dictionary of fvSolution that is often used in OpenFOAM is relaxationFactors which controls under-relaxation, a technique used for improving the stability of computation, particularly in solving steady-state problems. A relaxation factor of 0.5 is introduced in laminar model simulations of flow Re1000 and Re2500. Residuals for Ux and p are plotted using Gnuplot. The residual plot of both case setups points out that the residual values came down to the order of 10e-6 range or lower which is a pretty good thing. Solutions got converged in 773 and 1861 iterations for Re1000 and Re2500 respectively. The same fvSchemes and fvSolution parameters and functions have been used here in our analysis study for Re1000 and Re2500 from that of the OpenFOAM tutorial base case, we have picked. The simulation was set up for running up to 1500 iterations using simpleFoam solver with includFunc of yPlus.

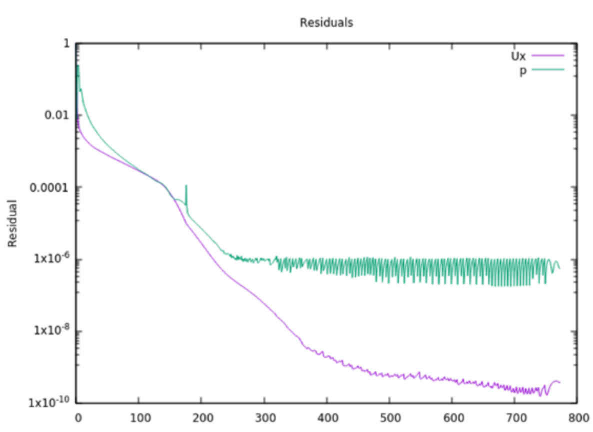

Fig 3.3.1.2.1 Residuals plot – Re1000

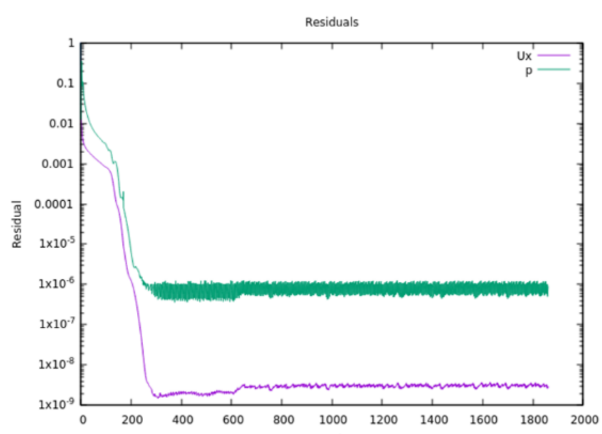

Fig 3.3.1.2.2 Residuals plot – Re2500

3.3.1.3 Post-processing

Post-processing gives users complete insight into their fluid simulation results. They are the most precise and quantitative way to present numerical data and a popular way of making a direct comparison between experimental and numerical data. It can also use as a primitive validation method to check the results from simulation. For example, we can plot the velocity profile at any point and time and it should be a parabolic profile with maximum velocity, double the value of average velocity. Umax value remains almost the same throughout the length of the pipe. Re1000 and Re2500 both follows the parabolic profile at inlet (fixedCodedValue) and outlet. It was found to be almost the same throughout the pipe.

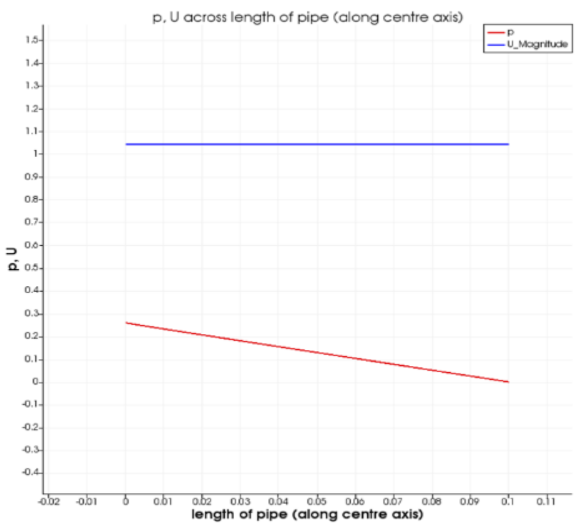

Fig 3.3.1.3.1 p,U across length of pipe – Re1000

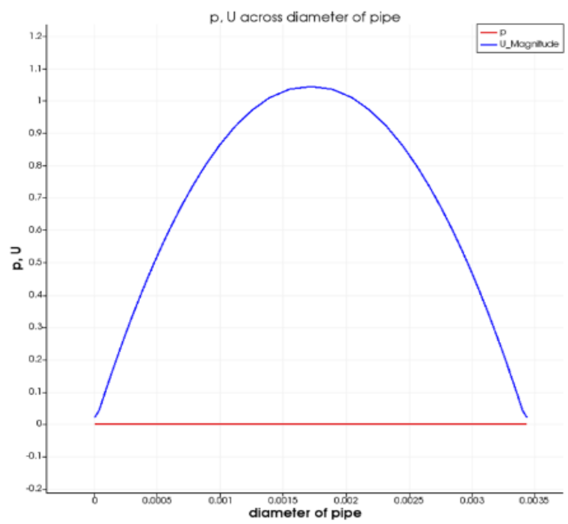

Fig 3.3.1.3.2 p,U across diameter of pipe – Re1000

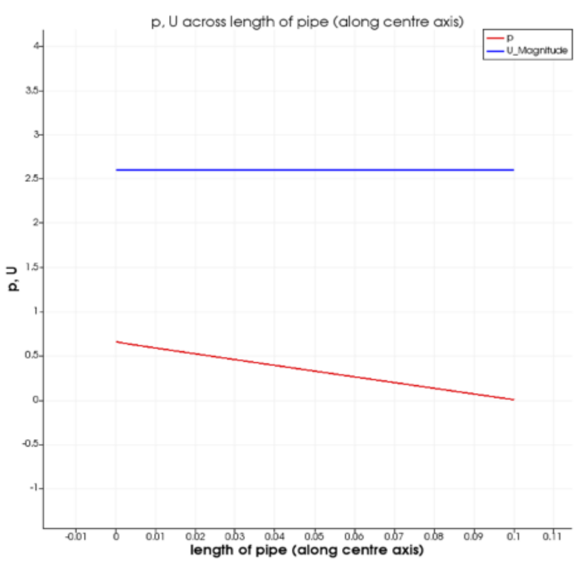

Fig 3.3.1.3.3 p,U across length of pipe – Re2500

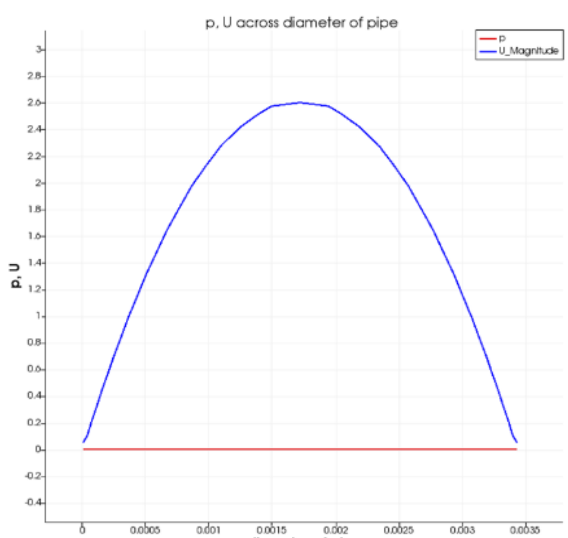

Fig 3.3.1.3.4 p,U across diameter of pipe – Re2500

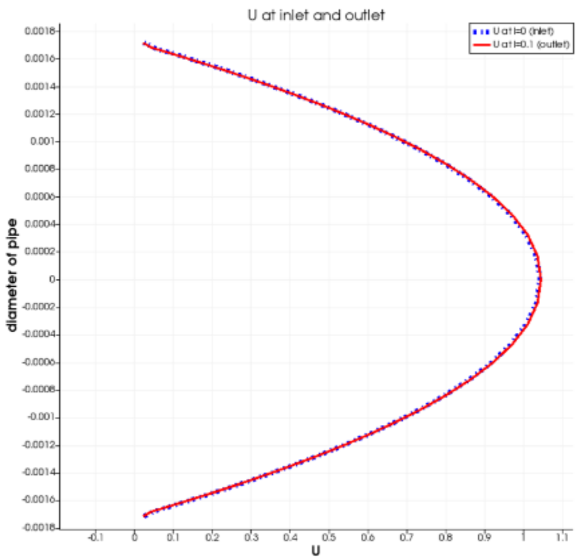

Fig 3.3.1.3.5 velocity profile inlet v/s outlet – Re1000

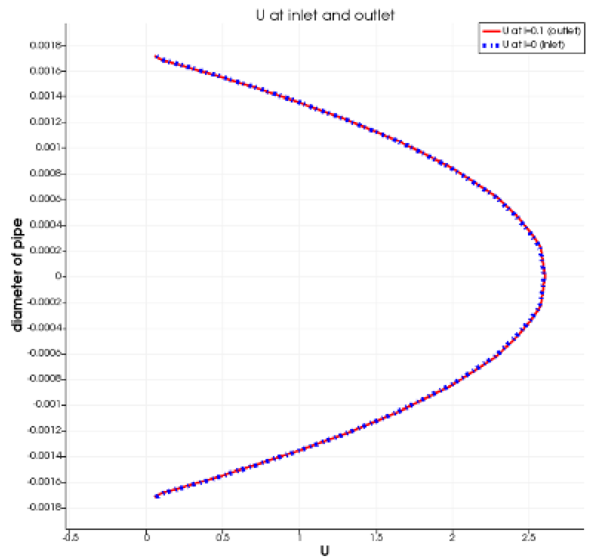

Fig 3.3.1.3.6 velocity profile inlet v/s outlet – Re2500

Contours can be fetched from simulation. Since we are running the same simulation setup using the same constraints by changing only the inlet velocity every time, contours are pretty the same in all simulations. Values differ, but the pattern remains the same. The general trend of velocity and pressure/density contours can be understood from the given below example of the Re1000 flow model. Pressure/density (p) was dropped from 0.26 m^2/s^2 to 0 (atmospheric pressure at the outlet). The maximum velocity value at the outlet was given to be 1.044m/s by simulation results, whereas the minimum velocity value at the nearby grid of pipe wall was found to be 0.021m/s.

Fig 3.3.1.3.7 pressure/density contour – Re1000

Fig 3.3.1.3.8 velocity magnitude contour – Re1000

Similarly, pressure/density dropped from 0.66 $m^2/_{s^2}$ to 0 for the Re2500 flow. The velocity profile took the exact parabolic profile. Umax value at the outlet was found to be 2.603m/s, which is two times the average velocity which is in good agreement with analytical solutions for the laminar models. Umin velocity value at the nearby grid of the pipe wall was found to be 0.052m/s.

3.3.2 Turbulent flow

Re3500, Re5000, and Re10000 were used to check the frictional losses in straight pipe turbulent flows. After checking on laminar flow, we checked these low Reynold's number fluid flows. Re5000 and Re10000 are clearly under a turbulent flow regime. Re3500 was checked against the turbulent flow to understand how well it is behaving like the turbulent flow. Frictional losses from these transitional and turbulent flows are studied. We started with Re3500 flow with the k-epsilon RANS model but switched to the k-omega SST model since the former failed to provide good results while the latter gave pretty good matching results.

3.3.2.1 Pre-processing & Boundary conditions

Re3500 – k-epsilon & k-omega SST

Re3500 flow belongs to the transitional flow regime. It makes no sense when we compare transitional flow with turbulent flow. But what we are trying to do here is to check how close is the transitional flow of Re3500 to the turbulent flow. Hence, we set up a case similar to the turbulent flow model. Initially, the k-epsilon RANS model was used to do the analysis study for this turbulent model. Since it failed to provide a good result with frictional losses, analysis was switched to the k-omega SST model, which turned out to be a good decision by giving pretty good matching results for the friction factor. A grid independence study was carried out and it followed similar patterns as before by giving better results for 2000 cell geometry with optimum computational cost.

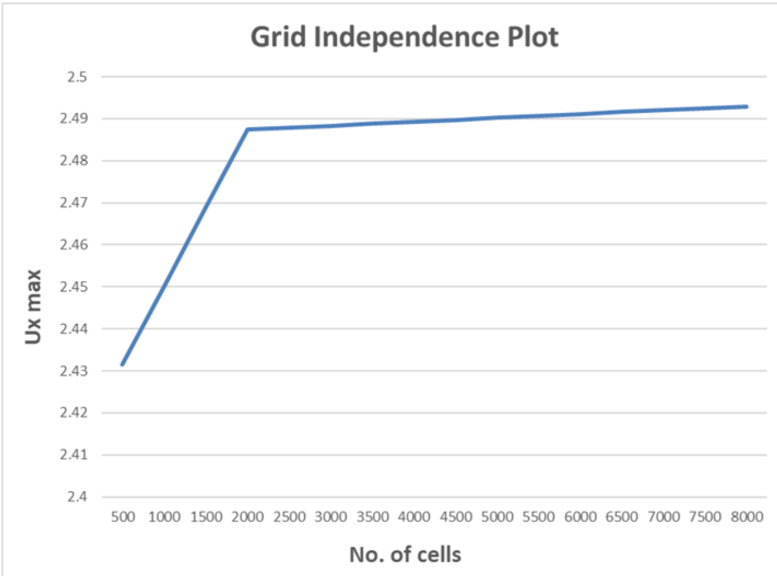

Fig 3.3.2.1.1 Grid independence plot – Re3500

No. of cells	Umax (m/s)
500 (50x10)	2.431
2000 (100x20)	2.487
8000 (200x40)	2.492

Fig 3.3.2.1.2 Grid independence table – Re3500

```
inlet
{
    type            codedFixedValue;
            value                   uniform (0 0 0);

            redirectType    parabolicVelocity;
            code
            #{
                    const fvPatch& boundaryPatch = patch();
                    const vectorField& Cf = boundaryPatch.Cf();
                    vectorField& field = *this;

                    const scalar R = 0.00172;
                    const scalar Umax = 2.289;

                    forAll(Cf, faceI)
                    {
                            const scalar r = Cf[faceI].z();

                            field[faceI] = vector(Umax*(pow((1-(r/R)),0.167)), 0 , 0);
                    }
            #};
}
```

Fig 3.3.2.1.3 Turbulent inlet velocity profile – Re3500

For easy convergence and steady-state flow, a turbulent velocity profile is applied here at the inlet as a boundary condition in the form of codedFixedValue. The maximum velocity was set to be 2.289. From Reynold's number equation, the average velocity which corresponds to the Re3500 flow was calculated. The maximum velocity was chosen from the empirical relation which states that 80% of the maximum velocity would be nearly the average velocity of the turbulent flow. Hence this value was applied to the codedFixedValue in the U directory. Nikuradse's empirical relation of the power-law profile for the turbulent flow was used to calculate the inlet turbulent velocity profile. From that equation, 1/n value was taken equal to 1/6, which corresponds to the turbulent flow of Re4000, which is considered to be the minimum Reynold number for a turbulent flow. Hence, we would be comparing the flow with the Re4000 turbulent model velocity profile and frictional losses. From validation methods, we would be able to fetch results and compare how close is the flow to the turbulent flow.

	inlet	outlet	axis	cylinder wall	front & back
p	zeroGradient	fixedValue uniform (0 0 0)	symmetryPlane	zeroGradient	wedge
U	codedFixedValue (max 2.289)	zeroGradient	symmetryPlane	noSlip	wedge
k	turbulentIntensity KineticEnergyInlet (intensity 0.057) (uniform 0.0167)	zeroGradient	symmetryPlane	kqRWallFunction (uniform 0.0167)	wedge
epsilon	turbulentMixing LengthDissipation RateInlet (mixing length 0.007) (uniform 0.051)	zeroGradient	symmetryPlane	epsilonWallFunction (uniform 0.051)	wedge
nut	zeroGradient	zeroGradient	symmetryPlane	nutKWallFunction (uniform 0.0005)	wedge

Fig 3.3.2.1.4 Re3500 k-epsilon boundary conditions

Appropriate boundary conditions were provided in all the dictionaries we were using for the k-epsilon model. The table mentioning boundary conditions used is added below. Later the same model was analyzed using the k-omega SST model. We used the same BC again except the epsilon directory was replaced by omega, which resulted in using omegaWallFunction instead of epsilonWallFunction at the cylindrical wall face. The equations used for calculating the values for k, epsilon, nut, and omega are given below :

$$\text{turbulent kinetic energy, } k = \frac{3}{2}\,(u_{avg}\, I)^2$$

$$\text{turbulent intensity, } I = 0.16\, Re^{-0.125}$$

$$\text{turbulent dissipation rate, } \varepsilon = \frac{C_\mu{}^{0.75}\, k^{1.5}}{0.07\, L}$$

$$\text{specific dissipation rate, } \omega = \frac{\varepsilon}{C_\mu k}$$

$$\text{turbulent viscosity, } nut = \frac{C_\mu k^2}{\varepsilon}$$

Re5000 & Re10000

The same initial setup and boundary conditions have been used for analyzing Re5000 and Re10000 flow models. Since the k-omega SST RANS model performed a good job with Re3500, the same turbulence model was used here again. Case setup directories and dictionaries were modified for Re5000 and Re10000 flows accordingly. Grid independence study was conducted for both Re5000 and Re10000 and still 2000 cell geometry offered better results with lesser computational cost. Similar to Re3500, a turbulent velocity profile was applied at the inlet as a BC. For Re5000, Nikuradse's power law profile empirical formula was applied. 1/n value was taken equal to 1/6, which was taken for Re3500 previously. 1/6 serves its function in the empirical relation for turbulent velocity profile for the low Reynold's number flows. By Nikuradse's equation, the 1/n value gets changed as Reynold's number increases. Hence for Re10000, the value was chosen differently. By the empirical relation proposed by Nikuradse, 1/n was taken as equal to 0.338 Re^-0.081 = 0.1603. This couldn't make much difference to the turbulent profile, since even Re10000 is considered as low Reynold's number itself.

3.3.2.2 Control parameters, fvSchemes & fvSolution

Re3500 – k-epsilon & k-omega SST

The same fvSchemes and fvSolution that we have used for laminar model analysis were used here initially for Re3500. It was observed that the solutions weren't converged until 3000 iterations. And it was also noted that the residuals were fluctuating heavily. The residual values were very unstable and oscillated from the very first to 3000 iterations. That is when we decided to study more about the divSchemes used in the fvScheme directory. Different divSchemes were applied to a 500 cell geometry and simulations were run with a relaxation factor of 0.9 applied. Including the previous divScheme, limitedLinear, a total of five divSchemes were tested. The residuals for Ux and p were plotted using Gnuplot, documented, and analyzed. Among them, linear, LUST, and upwind converged in 184, 171, and 132 iterations respectively, while limitedLinear and limitedLinear01 didn't. Residuals were very stable for limitedLinear01, yet failed to converge results. An interesting fact was that all the divSchemes gave pretty much close results for Umax and p value. Even though some didn't converge, all of the divSchemes got the residuals lowered to a point of order 10e-5 or more, which clearly says the error is pretty low. Residuals plot of the above-mentioned divSchemes can be found here :

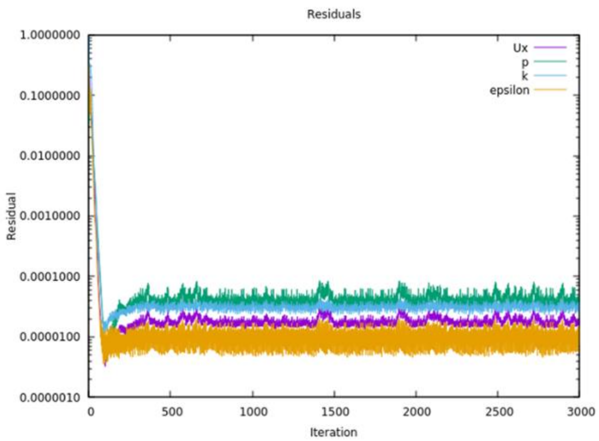

Fig 3.3.2.2.1 limitedLinear

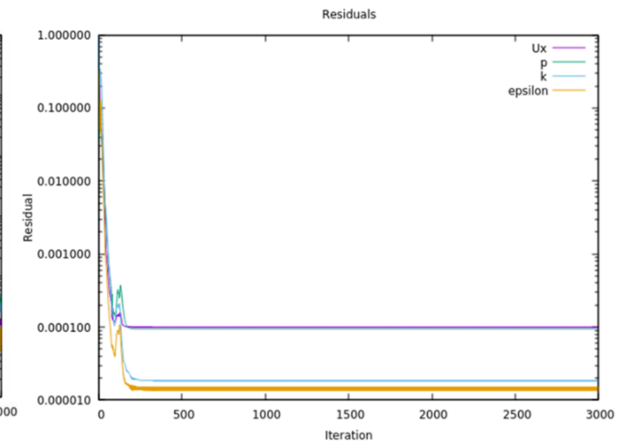

Fig 3.3.2.2.2 limitedLinear01

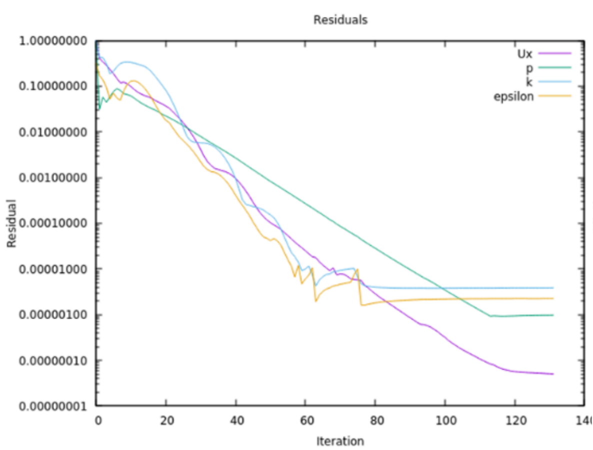

Fig 3.3.2.2.3 upwind

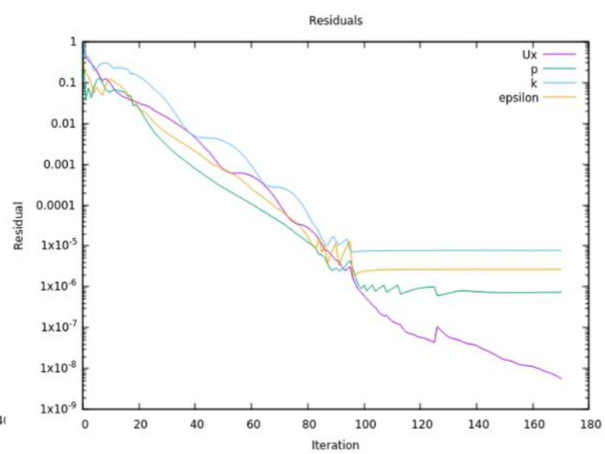

Fig 3.3.2.2.4 LUST

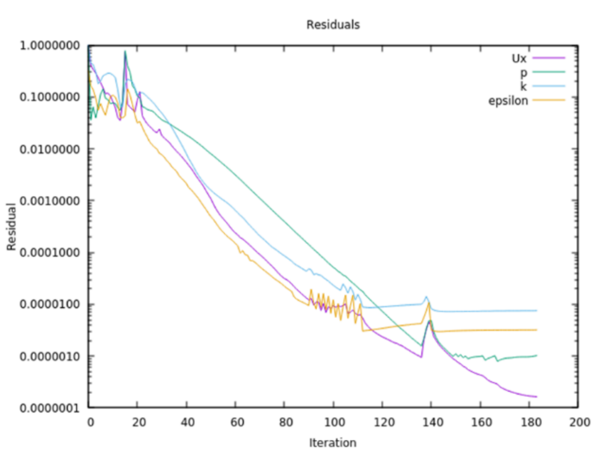

Fig 3.3.2.2.5 linear

Divergence schemes	**Umax at outlet (m/s)**	**p at inlet (m^2/s^2)**
linear (184)	2.43144	4.78395
LUST (171)	2.43138	4.78274
limitedLinear	2.43144	4.77933
limitedLinear01	2.4318	4.70961
Upwind (132)	2.43157	4.71148

Table 3.3.2.2.1 divSchemes – Umax, p values

linear is a second order, unbounded divScheme, which was taken for the further analysis for Re3500 flow. The divScheme was applied to the 2000 cell geometry with a relaxation factor of 0.9. Same divScheme was used for both the k-epsilon and k-omega SST RANS models. With 2000 cells, linear divScheme failed to converge for the k-epsilon model within 3000 iterations using simpleFoam. The residual values were very unstable and oscillating. But, with the k-omega SST model, linear divScheme gave pretty good convergence with 549 iterations. Residuals came down to the order of 10e5 to 7.

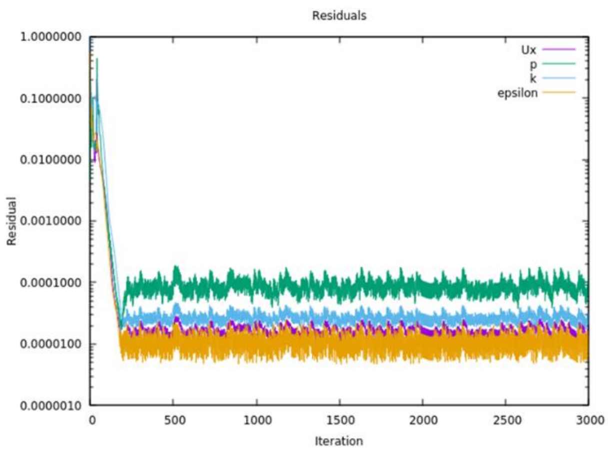

Fig 3.3.2.2.6 k-epsilon - linear

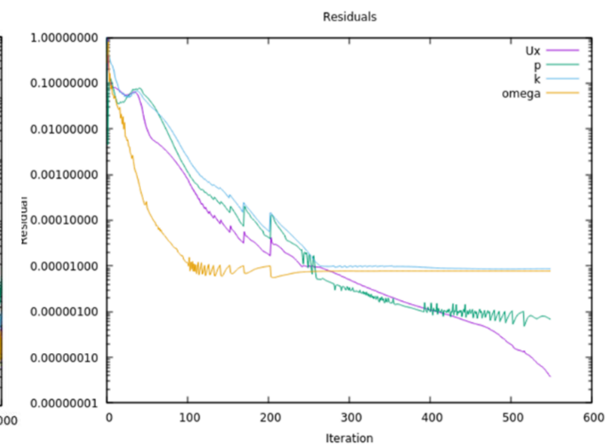

Fig 3.3.2.2.7 k-omega SST - linear

Re5000 & Re10000

2000 cell geometry with an applied relaxation factor of 0.9 was simulated for 3000s using the same fvSchemes and fvSolutions. Same BCs were applied, and residuals were plotted using Gnuplot. For Re5000, solutions converged with 2472 iterations and residuals came down to an order of 10e-6. But, Re10000 not only failed to converge the results within 3000 iterations but was also very unstable and oscillating. Hence, we switched back the divScheme used for the analysis from linear to limitedLinear, which was the one we started the analysis with. limitedLinear is a linear scheme that limits towards upwind in regions of rapidly changing gradient. Here, the solutions aren't yet converged but are very much steady and not oscillating at all. Residuals were dropped to an order of 10e-5, which is quite satisfactory.

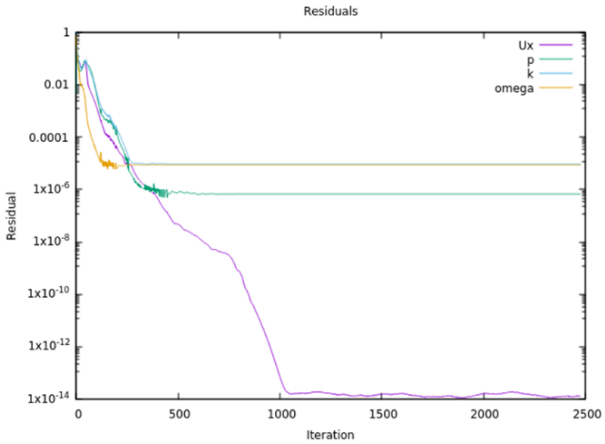

Fig 3.3.2.2.8 Re5000

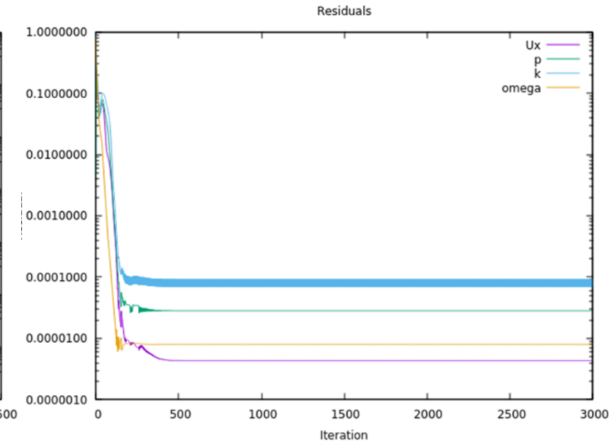

Fig 3.3.2.2.9 Re10000

3.3.2.3 Post-processing

Re3500 – k-epsilon & k-omega SST

Contours are pretty much the same as the ones we have already mentioned above (laminar). It follows the same pattern, only differs in values. Under the analysis using the k-epsilon model, the pressure/density value dropped from 5.14 $^{m^2}/_{s^2}$ to 0. The maximum velocity at the outlet was found to be 2.487m/s and the minimum to be 0.395 near the pipe wall. When we switched the model to the k-omega SST RANS model, values got changed noticeably. The maximum pressure/density value at the inlet was 2.379 $^{m^2}/_{s^2}$ and 0 at the outlet. Maximum and minimum velocity at the outlet and nearby pipe wall was noted as 2.444m/s and 0.17m/s respectively. It is important to note that pressure/density, maximum and minimum velocity values given by k-epsilon and k-omega SST models are different and will give entirely different pictures of the analysis study and result evaluation.

Velocity profiles at the outlet given by both k-epsilon and k-omega models are quite similar. Both models succeeded to give close matching Umax values, 2.487 and 2.444 by k-epsilon and k-omega SST models respectively. The velocity profile is far away from the empirical velocity profile calculated. This is because the flow, Re3500 is still in the transition phase and has not yet started the turbulent flow. Since the empirical formulae used is that of the turbulent flow of Re4000, the profile isn't a perfect match just like that in the laminar flow. In fact, this inaccurate match of Re3500 with turbulent flow profile can be considered as a success and valid result given by OpenFOAM.

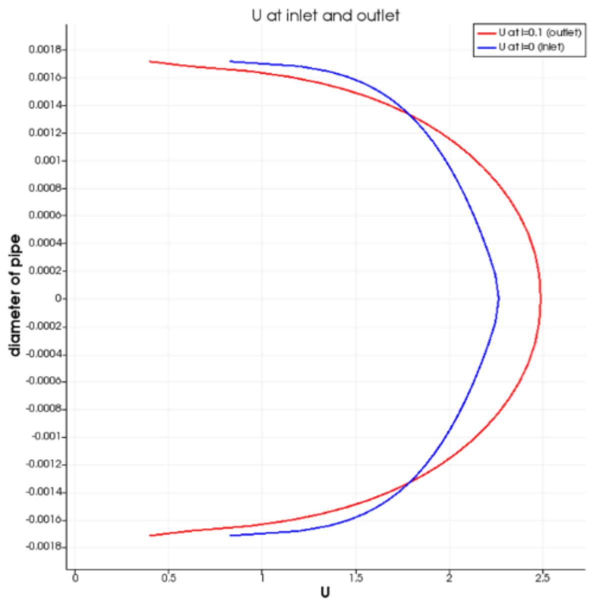

Fig 3.3.2.3.1 Velocity profiles – k-epsilon

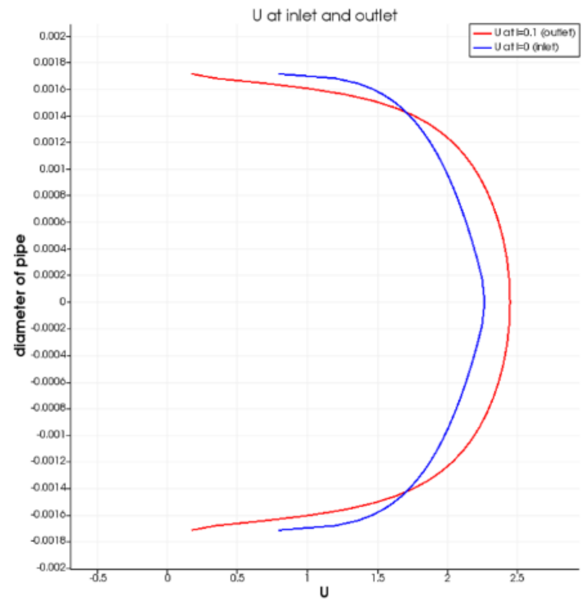

Fig 3.3.2.3.2 Velocity profiles – k-omega SST

Re5000 & Re10000

Results from Re5000 and Re10000 were quite satisfactory. Both models were successful to provide good velocity profile matches and other results. Contours were exactly the same as the expected pattern from the previous simulations. Pressure per unit density was 4.12 $^{m^2}/_{s^2}$ at the inlet dropped to 0 at the outlet for Re5000 where it was 12.88 $^{m^2}/_{s^2}$ for Re10000 flow. For both flows, maximum velocity was found at the outlet, Umax to be 3.396m/s and 6.513m/s for Re5000 and Re10000 respectively. Minimum velocity values, 0.29m/s and 0.965 were found at the nearby grid of pipe wall.

3.4 Rough Pipe

Rough pipes account for turbulent flow. Surface roughness enhances fluid viscosity at walls and thus turbulence is induced. Roughness height and distribution directly affect the friction factor of the pipe. This depends on the material, surface finish, Reynold's number, relative roughness, and so on.

3.4.1 Turbulent flow

Relative roughness and Reynold's number are the two major parameters that affect the friction factor in rough pipes. Empirical formulas and experimental results are out there to check the frictional losses and velocity profiles of turbulent flow. With the help of the Moody chart and Colebrook equation, we find the friction factor for turbulent flows in rough pipes. In our study, we will check the simulation results of Re5000 and Re10000 flows in a rough pipe.

3.4.1.1 Pre-processing & Boundary conditions

For the analysis study of rough pipe, stainless steel is taken as our pipe material, assumed to be having a uniform internal surface roughness with an average roughness height of 0.015mm. During the analysis study for both flows of Re5000 and Re10000, the rough pipe is considered to be the same one. Hence, apart from the surface roughness thing, all the other previous settings we had done for the smooth pipe are applied here again. Exactly the same geometric dimensions have been used. Hence wedge geometry with the same mesh settings was applied. During the grid independence study, it was evident that 2000 cell geometry was a better deal considering results accuracy and computational cost. Hence that was picked up for further analysis with the rough pipe as well. Boundary conditions were also applied similarly to that of the smooth pipe, the only exception was the wall function applied. nutKWallFunction in the nut directory was replaced by nutKRoughWallFunction. This specific one helped us to introduce the roughness parameters. Roughness height and distribution were defined during the analysis using this function. There are two parametric values to be defined, Ks and Cs. Ks, the average roughness height was taken as 0.015mm, considering stainless steel pipe. Cs value was taken as 0.5, which points out that the roughness is uniform throughout the pipe.

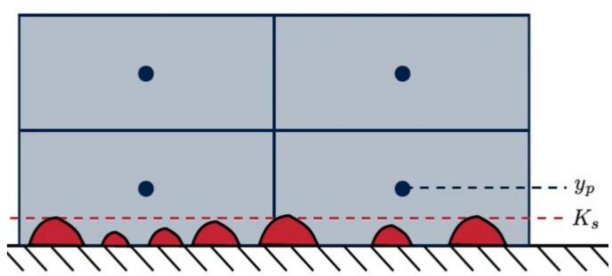

Fig 3.4.1.1.1 Roughness height – Rough pipe

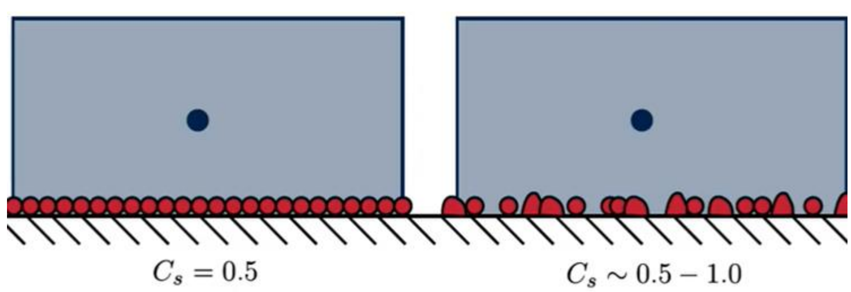

Fig 3.4.1.1.2 Roughness distribution – Rough pipe

```
{
    type            nutkRoughWallFunction;
    Ks              uniform 0.000015;
    Cs              uniform 0.5;
    value           $internalField;
}
```

Fig 3.4.1.1.3 nutKRoughWallFunction – Rough pipe

The pre-processing and boundary conditions remain the same for both cases of Re5000 and Re10000, except for the inlet velocity boundary condition applied. At inlet, max of 3.27m/s flow velocity was applied for Re5000, while Re10000 had 6.541m/s. A turbulent inlet velocity profile was applied to both flows just like that we had done for the respective flows in the smooth pipe.

3.4.1.2 Control parameters, fvSchemes & fvSolution

The same fvSchemes and fvSolution have been used here for both flows as that for the flows in the smooth pipe. This could facilitate further comparative studies. Hence, the case set up for Re5000 flow was simulated with linear divScheme with a relaxation factor of 0.9. On the other hand, Re10000 was run for 5000 iterations with limitedLinear divScheme with the same relaxation factor. For the flow of Re5000, the solutions appeared to be converged with 3235 iterations. Residual values came down in a pretty good way. Solutions didn't converge within the specified iterations for the flow of Re10000. The residuals are very stable and low in the order of 10e-4,5 and 6. Hence the results can be considered as good and satisfactory. Residual values for p, Ux, k, and omega were plotted using Gnuplot.

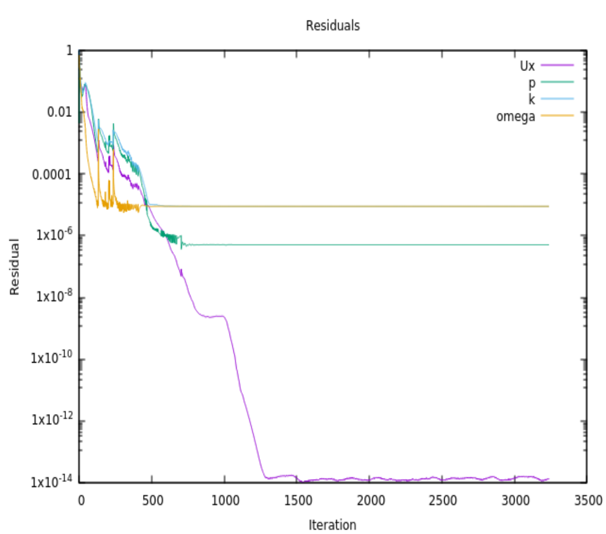

Fig 3.4.1.2.1 Residuals plot – Re5000 Rough pipe

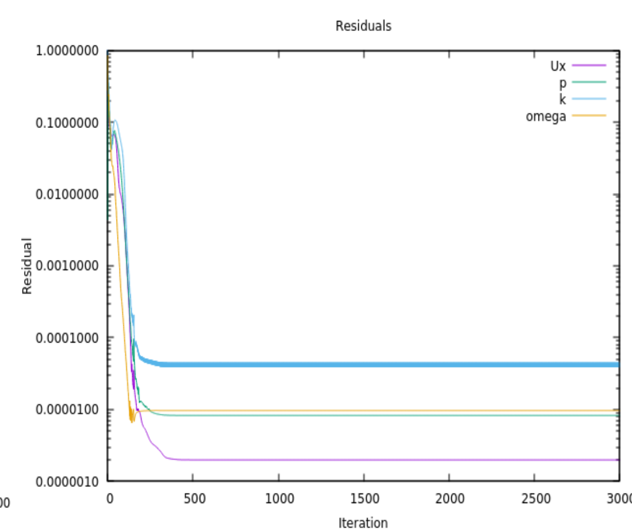

Fig 3.4.1.2.2 Residuals plot – Re10000 Rough pipe

3.4.1.3 Post-processing

Results from the analysis of Re5000 and Re10000 seem quite satisfactory. Contours were following the same pattern as that of the previous analysis. The pressure/ density value dropped from 4.12 $^{m^2}/_{s^2}$ at the inlet to 0 $^{m^2}/_{s^2}$ at the outlet for Re5000. The maximum velocity value was found to be 3.39m/s at the center of the pipe while the minimum velocity value at the first grid near the pipe wall was 0.29m/s. The analysis results of Re10000 show that the pressure/density value at the inlet was 12.88 $^{m^2}/_{s^2}$ and dropped to 0 $^{m^2}/_{s^2}$ at the outlet. The maximum velocity was 6.513m/s at the pipe center and the minimum velocity was 0.965m/s at nearby the pipe wall. Different graphs were plotted for both Re5000 and Re10000 and exported for analysis and comparative study. These can be further used for validation purposes as well.

Chapter 4

Results

4.1 Smooth Pipe

4.1.1 Laminar flow

Laminar flow results can be compared / validated against analytical solutions, we have. In this way, we could know how accurate are our simulation results. Results are organized in the form of tables and graphs.

4.1.1.1 Analytical Results

Laminar flow in smooth pipes has analytical solutions for defining its velocity profile, pressure drop, and friction factor. Often these analytical solutions are used as validation methods to check the numerical results provided by the simulation model. The velocity profile of laminar flow follows a parabolic profile where the maximum velocity is approximately double the average velocity of the flow. The analytical equation is the same for both the Re1000 and Re2500 models, we have used. This is under the circumstance that Re2500 is still in its laminar flow regime, not yet entered into the transitional phase.

From the analytical equation, the maximum velocity value for the laminar flow of R1000 should be 1.046m/s. This value is calculated from the average velocity value, 0.523m/s which was obtained from Reynold's number equation. Similarly, Re2500 flow should have a Umax value of 2.616m/s derived from the Uavg value, 1.308m/s. From the analytical equation, the friction factor can be calculated and found to be 0.064 and 0.0256 for Re1000 and Re2500 respectively.

4.1.1.2 Numerical Results

From the OpenFOAM simulation results and further calculations, it is possible to find the maximum velocity, velocity profile, pressure drop, friction factor, and so on. The friction factor is representing the loss of pressure of a fluid in a pipe due to the interactions between the fluid and the pipe, on which this project work is mainly focused. Umax value was found to be 1.044m/s at the center of the pipe which is in good agreement with the analytical value. The velocity profile can be plotted separately and validated against the analytical velocity profile graph. Contours provide detailed results about velocity and pressure at any point in the pipe at any time. Pressure drop can be thus calculated using numerical calculations. From the equations, head loss in the pipe was found to be 0.02643m. From the Darcy-Weisbach equation, the friction factor by simulation was calculated as 0.0635.

For the flow of Re2500, the maximum velocity found was 2.603m/s by numerical analysis. The velocity profile can be spotted and validated against the analytical profile. Maximum velocity is found at the pipe center while the pipe wall is having minimum velocity with a parabolic profile overall. From numerical calculations using pressure drop, the head loss in the pipe was found to be 0.06674m, which in turn provided a friction factor of 0.0252 from the Darcy-Weisbach equation.

4.1.1.3 Result validation

Result comparison and validation is the most crucial part of any analysis study. Without validation, it is impossible to ensure that we have done a good simulation or obtained accurate results for the fluid problem. Verification and validation examine the errors in the simulation results. Credibility is obtained by demonstrating acceptable levels of uncertainty and errors. These validations of numerical results can be performed against analytical / empirical / experimental results. The percentage error says how close is the simulation results. For the laminar flow model, the results from contours can be used as another validation method. This equation connects the pressure difference to the average velocity value.

$$P1 - P2 = \frac{32\ \mu u_{avg} l}{d^2}$$

Hence the pressure contour result from the numerical simulation is used in the equation to check the approximate average velocity of the flow. We could check whether this value is nearby the analytical average velocity value, 0.523m/s which was received from the Reynolds number equation. From the pressure difference between inlet and outlet, the average velocity was found to be 0.532m/s which is a good result. Likely, for Re2500, the Uavg value was found to be 1.34m/s which is a good match with the analytical result of 1.308m/s.

The velocity profile at the pipe outlet of the flows Re1000 and Re2500 was plotted against the analytical velocity profile equation. The result was very much satisfying and found a perfect match in profiles. Thus, it made clear that OpenFOAM is doing a great job in laminar modeling and analysis. Numerical and analytical profiles for both Re1000 and Re2500 take a perfect parabolic profile with Umax value as mentioned previously. These profiles seem very much identical in shape and only differ in numerical values.

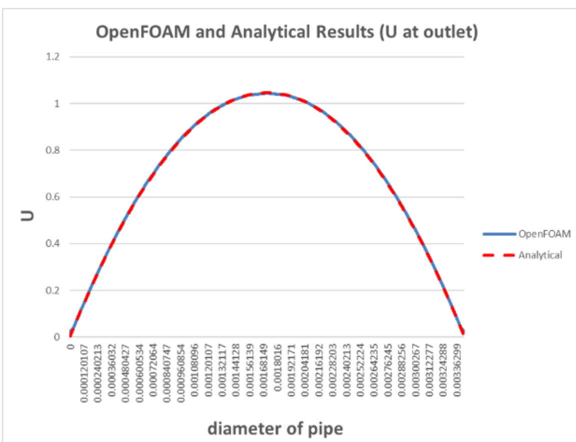

Fig 4.1.1.3.1 Velocity profiles – Re1000

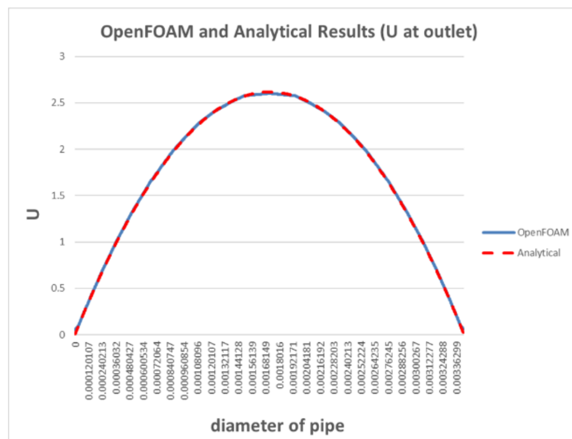

Fig 4.1.1.3.2 Velocity profiles – Re2500

The result comparison and percentage error between the analytical and numerical results in terms of maximum velocity and friction factor for both Re1000 and Re2500 are added here in table format for easy reference:

	Parameters	**Analytical Value**	**OpenFOAM value**	**Percentage error**
Re1000	Umax	1.046	1.044	0.19%
	Friction Factor	0.064	0.0635	0.78%
Re2500	Umax	2.616	2.603	0.49%
	Friction Factor	0.0256	0.0252	1.56%

Table 4.1.1.3.1 Results comparison – Re1000, Re2500

4.1.2 Turbulent flow

Re3500 (transitional flow) is compared against the turbulent flow and likeness is noted. Low Reynold's number turbulent flows of Re5000 and Re10000 are compared and validated against the empirical formulas and Moody chart. Results are recorded, organized, and presented through graphs and tables.

4.1.2.1 Empirical Results

Re3500 – k-epsilon & k-omega SST

Re2500, a transitional range flow was designed and analyzed using the laminar model and found good agreement with laminar flow analytical results. This stated that R2500 still behaves like laminar flow. Similarly, Re3500, which falls under the transitional flow regime, was analyzed using the turbulent model to check how close is it to the turbulent flow. There is no actual method exists to check the nature of the transitional flow. The transitional flow regime is a field where high research and study are still going on. Since we benchmarked the turbulent flow here, the empirical formulae used was that of the closest turbulent flow, say Re4000. Nikuradse's empirical power law velocity profile equation was used to check the velocity profile. 1/n value from the equation was chosen as 1/6, which corresponds to the Re4000 turbulent flow. These are not analytical, but empirical results. Empirical relations derived from experimental results are used for analysis and validation. Hence, it is true that we cannot expect a perfect match in these results. The difference in profiles points out how close is our results to the experimental results. Here, through the empirical relation given, the maximum velocity for the flow (Re4000) is calculated as 2.289m/s. The empirical value for maximum velocity is calculated in such a way that 80% of the Umax value must be the average velocity value obtained directly from Reynold's number equation. That's how the average velocity and maximum velocity are connected through empirical relations for the turbulent flow. From the empirical relation, the predicted friction factor for the smooth pipe with a turbulent flow of Reynold's number 3500 is calculated as 0.0411. Apart from the empirical equation, the Moody chart is another way to calculate the friction factor of the flow in the pipe. Here, we check the friction factor value for the smooth pipe (surface roughness height taken as 0) corresponding to particular Reynold's number. It was found to be around 0.0413 which is in very good agreement with the empirical relation.

Re5000 & Re10000

Coming to the turbulent flows of Re5000 and Re10000, Nikuradse's empirical power law velocity profile equation is used for both. For Re5000, 1/n value was taken as 1/6 which was mentioned for the very low Reynold's number turbulent flows from Re4000. From Reynold's number equation, the average velocity was calculated as 2.616m/s. Maximum velocity was calculated as 3.27m/s from the average velocity value, dividing by 0.8. Similarly, for Re10000 calculations did and empirical relations took similar to that of the case of Re5000 except for the selection of 1/n value. Even though 1/n can be taken as 1/6 for flows below Re23000, according to Nikuradse's empirical formulation, a general trend equation proposed by himself is used here. According to the equation, 0.1603 is taken as the 1/n value. From the empirical equations, the average velocity was found to be 5.233m/s, which gave out, the maximum velocity to be 6.541m/s. Calculations are done in such a way that 80% of Umax is Uavg. From the empirical equations for the turbulent flow, the friction factor corresponding to Re5000 and Re10000 are 0.0376 and 0.0316 respectively. Moody chart offers very similar results for both flows in the smooth pipe, 0.0375 and 0.0315 respectively. Moody chart value and empirical relations are in good terms giving out pretty much similar results for the friction factor.

4.1.2.2 Numerical Results

Re3500 – k-epsilon & k-omega SST

The numerical simulation study for Re3500 was carried out by the k-epsilon RANS model initially. The graphs were plotted and the results were compared with the known empirical results. Umax value was a good match, but the friction factor wasn't. That's when it was decided to check the flow with the k-omega SST RANS model. This time both the results, velocity, and friction factor were good matches. The maximum velocity values delivered by the k-epsilon and k-omega models were 2.487m/s and 2.444m/s respectively. K-epsilon and k-omega SST models performed a good job in delivering details about the velocity profile greatly. Considering the velocity profiles at the outlet of the pipe, it is evident that both models predicted the profile somewhat similar only. If we need to pick one better than the other, the k-omega SST model predicted the profile better. The profile is a bit flatter than that of the k-epsilon model and the Umax point came closer to the empirical profile point. When we consider the friction factor delivered by both models, there comes a gap. From the pressure contour results, the head loss was calculated as 0.524m by k-epsilon and 0.242m by the k-omega SST model. That is a difference of more than 100% between these two models which is not at all acceptable. From the Darcy-Weisbach equation, using the pressure drop and head loss values, the friction factor given by k-epsilon was 0.089 and 0.0428 by the k-omega SST model. A huge difference between these values is clearly visible.

Re5000 & Re10000

Analyzing simulation results of Re5000, the k-omega SST model gave a maximum velocity value of around 3.39m/s. Pressure contours point out that the head loss for Re5000 under the simulation constraints was 0.42m. Applying the head loss in the Darcy-Weisbach equation, the friction factor turns out to be around 0.0385. Likely, Re10000 was also modeled and analyzed using k-omega SST RANS. The maximum value was found to be 6.513m/s at the center of the pipe at the outlet. Calculations carried out from pressure contours indicated 1.313m head loss, which was applied in the Darcy-Weisbach equation for calculating the friction factor. The friction factor for this case, Re10000 was numerically found to be around 0.326. The velocity profiles for Re5000 and Re10000 were plotted at the outlet of the pipe and it was clear that the profiles become flattered as Reynold's number increased. With the increase in Reynold's number, the flow is getting more turbulent and fully developed, which in turn changes the flow profile into flatter in the middle and sudden drop at walls.

4.1.2.3 Result validation / comparison

Re3500 – k-epsilon & k-omega SST

The maximum velocity value given by both k-epsilon and k-omega SST models was a good match and the percentage error stays within a limit of 10% only pointing out that both models were successful to predict the maximum flow velocity for turbulent flow in smooth pipes. The velocity profiles given by both models were compared against the empirical profile separately and validated. It is clear that both models failed to replicate the exact empirical profile, but gave a close match. The reason behind this is that the flow Re3500 is not yet transformed into a turbulent one, but is still under the transitional phase only. The empirical profile was formulated according to the lowest possible turbulent flow, by considering the Re4000 model. So, comparing the velocity profile of Re3500 with the empirical profile of Re4000 must show deviations, which happened in our case is a good symbol of possible good results by OpenFOAM turbulent models. The flow is coming down from the laminar profile to the turbulent one. This is the reason why the maximum velocity points are above the predicted empirical profile. If we compare the profiles between k-epsilon and k-omega SST, both performed a similar job. However, the k-omega SST model profile appears to be a bit flatter than the k-epsilon model.

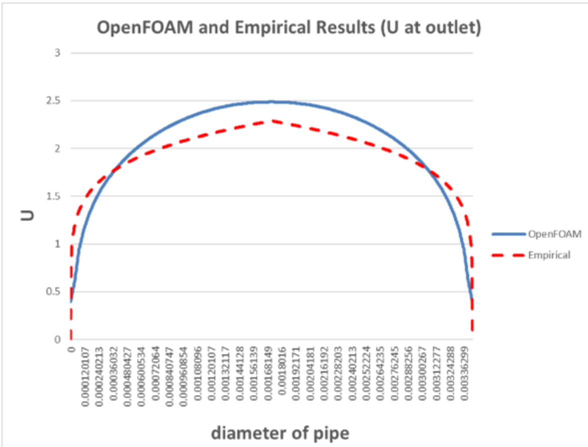

Fig 4.1.2.3.1 Velocity profiles – Re3500 k-epsilon

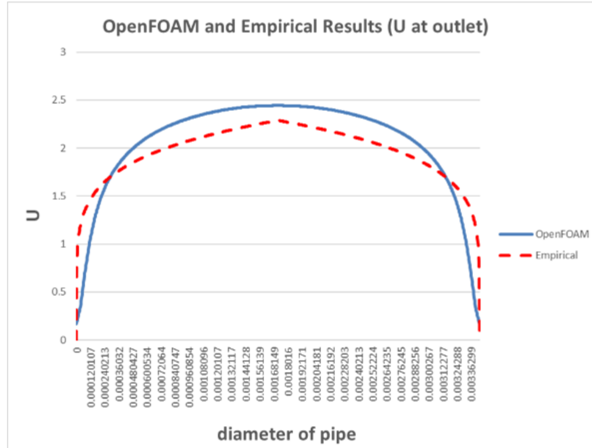

Fig 4.1.2.3.2 Velocity profiles – Re3500 k-omega SST

Considering the frictional factor derived from both models, the k-omega SST model gave a pretty good match with the empirical results and Moody chart value, while the k-epsilon model utterly failed to predict the frictional factor value for Re3500. For the k-epsilon model, even though it provided good results for Umax and velocity profile, the percentage error for friction factor is more than 100% which makes this model not at all acceptable for the frictional loss study. On the other hand, the k-omega SST model gave accurate results on Umax value, a better velocity profile, and a very good match for friction factor value. Hence, the k-omega SST model was used for the rest of the flow model analysis study. Parametric values and their percentage errors are given in tabular form here :

Re3500	Parameters	Empirical Value	OpenFOAM value	Percentage error
K-epsilon	Umax	2.289	2.487	8.65%
	Friction Factor	0.041	0.089	117.07%
K-omega SST	Umax	2.289	2.444	6.76%
	Friction Factor	0.041	0.043	3.63%

Table 4.1.2.3.1 Results comparison – Re3500

Re5000 & Re10000

Comparing numerical results with the empirical and Moody chart values for the friction factors of turbulent flows of Re5000 and Re10000, it is evident that the k-omega SST model is working very well to provide better and more accurate results. It is highly satisfactory that the results are within a percentage error of 3 and 4% maximum. The velocity profiles obtained from the simulation results are plotted against the empirical profile and found a better match than the previous one, Re3500. This is because the flow is getting more turbulent and fully developed as Reynold's number is getting increased. Hence, the pattern is now clear that with the increase in Reynold's number, the velocity profile is getting a better match with the empirical plot of the velocity profile. That says the profile of Re10000 is more coinciding with the empirical velocity plot than that of the Re5000. Hence it can be assumed that with a higher Reynold's number, the model would be able to give more perfect matching results for the velocity profile. Umax values given by both Re5000 and Re10000 models were also almost equal to the empirical values and within a percentage error of 4% and 1% respectively, which was quite satisfactory.

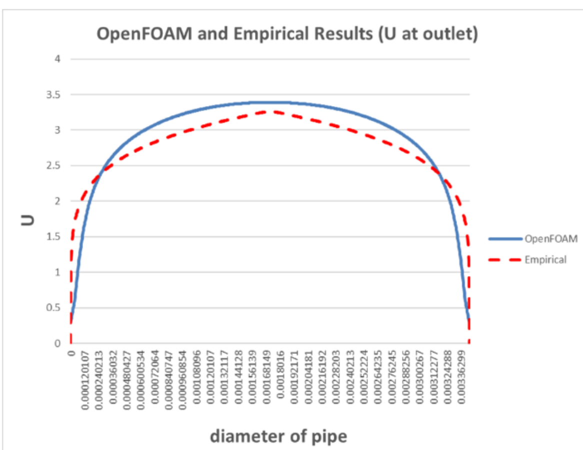

Fig 4.1.2.3.3 Velocity profiles – Re5000

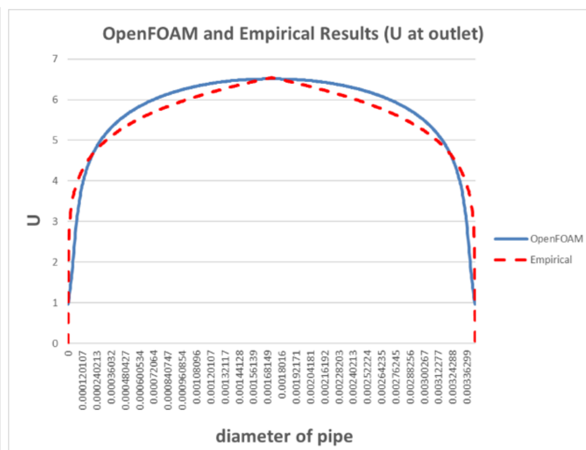

Fig 4.1.2.3.4 Velocity profiles – Re10000

	Parameters	**Empirical Value**	**OpenFOAM value**	**Percentage error**
Re5000	Umax	3.27	3.39	3.67%
	Friction Factor	0.0376	0.0385	2.39%
Re10000	Umax	6.541	6.513	0.43%
	Friction Factor	0.0316	0.0326	3.16%

Table 4.1.2.3.2 Results comparison – Re5000, Re10000

The parametric values for Re5000 and Re10000 with a comparative percentage error table are added above for easy reference.

4.2 Rough Pipe

4.2.1 Turbulent flow

Turbulent flow results can be compared / validated against empirical / experimental solutions, we have. In this way, we could know how accurate are our simulation results. Results are organized in the form of tables and graphs.

4.2.1.1 Analytical / Empirical Results

For the turbulent flow in the rough pipe, Nikuradse's empirical relation of power-law velocity profile was used. Using this power law equation, velocity profile at any point can be plotted and maximum velocity can be found. These data are extracted from the empirical relation, for both flows of Re5000 and Re10000. The empirical velocity profile was applied for both flows at the inlet as a boundary condition for the U directory. This was done by assuming the ideal turbulent flow at the inlet itself. Thus, the flow becomes steady state easily and helps in converging the solution early. Similar to that of the smooth pipe, the equation takes 1/6 as the value for 1/n for Re5000 flow, while for Re10000, 1/n is taken as 0.1603 (0.338Re^-0.081). The maximum velocity value, according to the empirical relation was found to be 3.27m/s and 6.541m/s for Re5000 and Re10000 respectively. These maximum values were found empirically from another relation, which connects Uavg with Umax. Values were taken such that 80% of the maximum value is the average value in turbulent flows. Hence, 2.616m/s and 5.233m/s are the average velocity values found in Reynold's number equation.

Results from post-processing were used to calculate the friction factor empirically. Colebrook equation was used to calculate the friction factor for rough pipe flows. Colebrook equation is an iterative solution approach so that we have to try multiple values through numerous iterations until we get a close match. What-If analysis in MS Office Excel was used here to find this iterative solution result. The friction factor value was found by equalizing the LHS and RHS of the Colebrook equation closely. And the value was found to be around 0.042 for Re5000 and 0.0368 for Re10000.

Friction Factor	LHS	RHS	LHS-RHS		Pipe Roughness	0.000015
0.041980458	4.88063596	4.8805258	0.000110162		Diameter	0.00344
					Relative Roughness	0.004360465
					Reynold nmbr	5000

Fig 4.2.1.1.1 Colebrook equation - Iterative solution from MS Excel – Re5000

Friction Factor	LHS	RHS	LHS-RHS		Pipe Roughness	0.000015
0.03685494	5.2089735	5.20901297	-3.94702E-05		Diameter	0.00344
					Relative Roughness	0.004360465
					Reynold nmbr	10000

Fig 4.2.1.1.2 Colebrook equation - Iterative solution from MS Excel – Re10000

Moody chart values were in good agreement with the results of the Colebrook equation. Hence the empirical results are validated from one another for these turbulent flows in the rough pipe. The friction factor values found for Re5000 and Re10000 from the Moody Chart are 0.042 and 0.0365 respectively.

4.2.1.2 Numerical Results

Numerical analysis for turbulent flows in the rough pipe was done by using the k-omega SST RANS model in OpenFOAM. The results were extracted in post-processing by Paraview and results were presented in the form of tables and graphs. The maximum velocity from the numerical analysis was found to be 3.39m/s for Re5000 and 6.513m/s for Re10000. Pressure/density values were taken from contours and head loss was calculated. It was found to be 0.42m and 1.313m respectively for flows of Re5000 and Re10000. Further calculations using head loss in the Darcy-Weisbach equation, the friction factor from the numerical analysis was found to be 0.0385 and 0.0326 respectively. The values are not very much satisfactory since they failed to show some significant progress in friction factor values from that of the smooth pipes. The values are only different from that of the smooth pipe when it comes to the third or fourth decimal points. The velocity profiles are getting flattered when it moves from Re5000 to Re10000. This indicates that with the increase in Reynold's number, the profile is getting a much better match with the empirical profile. The profile shows the maximum velocity value at the center of the pipe with a flatter profile, while the velocity value gets a sudden drop when it comes near the pipe wall. The velocity profile, maximum velocity value, and friction factor values obtained from numerical simulation using the k-omega SST model in OpenFOAM can be compared / validated against the empirical results. This could indicate how close are our numerical results with the empirical ones.

4.2.1.3 Result validation / comparison

Results and graphs received from the numerical analysis can be compared / validated against the empirical solutions we have. This would help us to understand how good is our results and how close we are. This validation and comparison help us to question what went wrong and extend our studies further. Else, we could prove that our numerical analysis perfectly supports the empirical results. In our case, the turbulent flows, Re5000 and Re10000 in the rough pipe were simulated using the k-omega SST model in OpenFOAM. Empirical solutions were found using the Colebrook equation and Moody Chart. The numerical result was validated against the empirical data. For, Re5000 the velocity profile is getting a good match while Re10000 is doing a better job. For Re10000, the velocity profile is flatter and coincides with the empirical velocity profile. This indicates that with the increase in Reynold's number, the velocity profile is getting a better match with the empirical profile. Considering the maximum velocity value, the percentage error between numerical and empirical values for both Re5000 and Re10000 are only within 1%. This was so impressive.

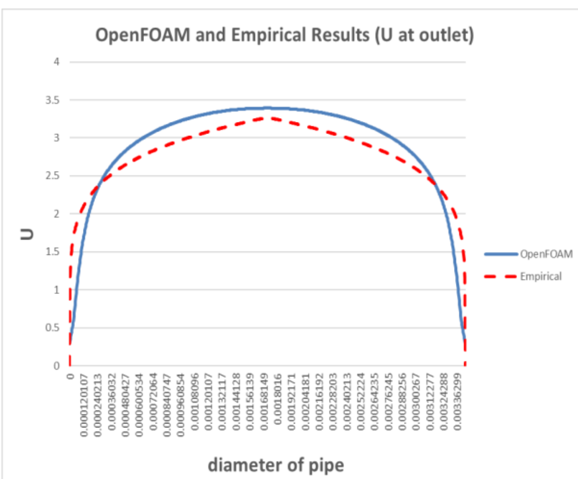

Fig 4.2.1.3.1 Velocity profiles – Re5000

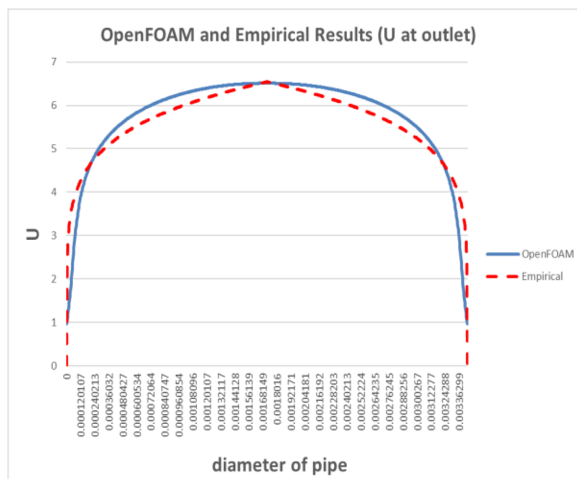

Fig 4.2.1.3.2 Velocity profiles – Re10000

Rough Pipe	Parameters	Empirical Value	OpenFOAM value	Percentage error
Re5000	Umax	3.37	3.39	0.59%
	Friction Factor	0.042	0.0385	8.23%
Re10000	Umax	6.541	6.513	0.43%
	Friction Factor	0.0365	0.0326	10.54%

Table 4.2.1.3.1 Results comparison – Re5000, Re10000

But, considering the frictional factor values, the things were not so impressive. For both cases, the percentage errors were around 10%, say 8.23% and 10.54% accurately for Re5000 and Re10000 respectively. The error percentage is within 10% which is somewhat okay, but not so satisfactory. The important thing to note here is that the frictional factor doesn't make much progress from that of the value we obtained from the smooth pipe. Obviously, the value is not exactly the same, but only differs when it comes to the third or fourth decimal points only. When we compare the results of the rough pipe with that of the smooth pipe, the pressure/ density value, maximum velocity, minimum velocity, velocity profile, and friction factor are very much alike. This might be because the surface roughness height value, Ks applied is very negligible (0.000015m), considering that of the smooth pipe (0). This might be the reason why rough pipe analysis delivers a result somewhat similar to that of the smooth pipe only. This is the case similar to both flows of Re5000 and Re10000. The comparison results are organized in graphs and tables, given below :

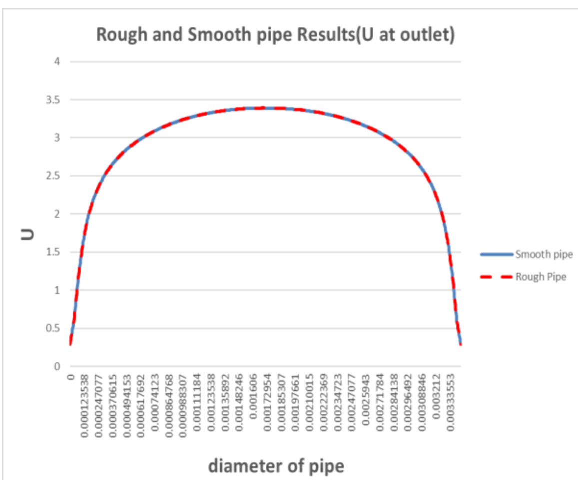

Fig 4.2.1.3.3 Velocity profiles – Smooth and Rough pipes - Re5000

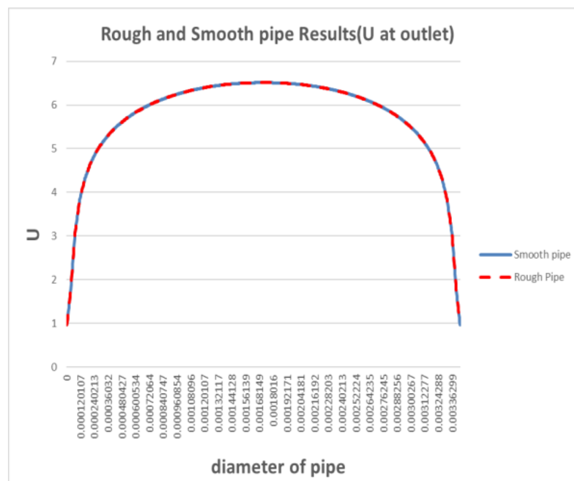

Fig 4.2.1.3.4 Velocity profiles – Smooth and Rough pipes – Re10000

Parameters	Rough Pipe	Smooth Pipe
p (m^2/s^2)	4.12558	4.12491
Umax (m/s)	3.39232	3.39066
Umin (m/s)	0.2956	0.29

Table 4.2.1.3.2 Parametric values – Smooth and Rough pipes - Re5000

Parameters	Rough Pipe	Smooth Pipe
p (m^2/s^2)	12.8791	12.8796
Umax (m/s)	6.51279	6.51331
Umin (m/s)	0.964831	0.9648

Table 4.2.1.3.3 Parametric values – Smooth and Rough pipes - Re5000

By comparing the results of smooth and rough pipes, it is evident that the results didn't move much. The implementation of nutKRoughWallFunction in the nut by replacing nutKWallFunction was the major change made in the simulation of rough pipe. In a nutshell, we could say that Ks and Cs are the only implementations made. Hence by varying these values, we could observe and study the change in analysis results, especially the friction factor. Since Cs is the distribution of surface roughness along the pipe, we would focus on the Ks value, which is the average surface roughness height. We could do a relation study between Ks and friction factor for different Ks values. Simulation can be run and friction factor can be noted by varying Ks value in each simulation. This could reveal the relation between Ks value and the friction factor. Thus, we could find any pattern or trend, if any such exists. Friction factor was found for different Ks values and was recorded in a tabular format and a relation graph was plotted. This was done for both turbulent flows of Re5000 and Re10000 and added below.

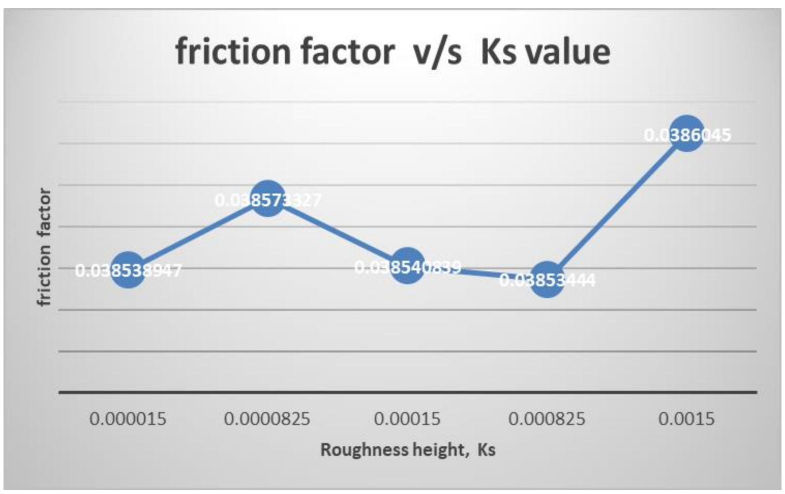

Fig 4.2.1.3.5 Ks v/s Friction factor – Re5000

Ks (m)	Friction Factor
0 (smooth pipe)	0.0385704
0.000015	0.0385389
0.0000825	0.0385733
0.00015	0.0385408
0.000825	0.0385344
0.0015	0.0386045

Table 4.2.1.3.4 Ks v/s Friction factor – Re5000

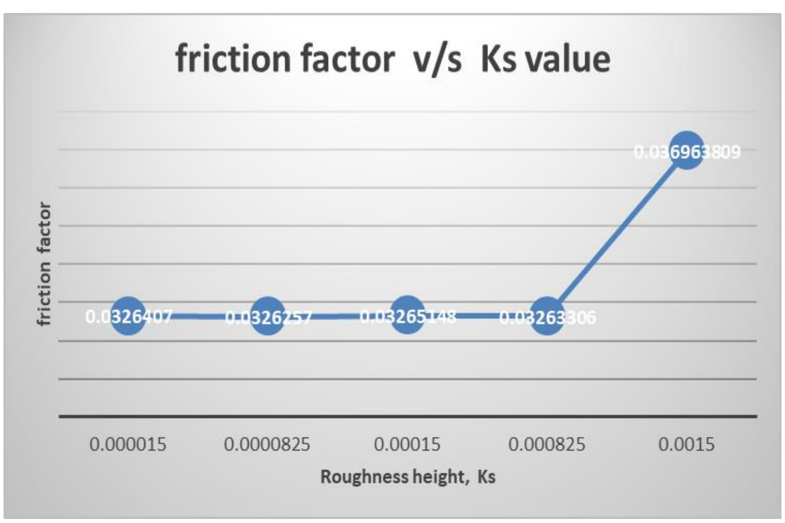

Fig 4.2.1.3.6 Ks v/s Friction factor – Re10000

Ks (m)	Friction Factor
0 (smooth pipe)	0.0326367
0.000015	0.0326407
0.0000825	0.0326257
0.00015	0.0326515
0.000825	0.0326331
0.0015	0.0369638

Table 4.2.1.3.5 Ks v/s Friction factor – Re10000

Chapter 5

Observation

We have done analysis for five flows, Re1000, Re2500, Re3500, Re5000 and Re10000. Re1000 was simulated under the laminar model. Re2500 and Re3500 were belonging to the transitional flow regime, but they were analyzed using laminar and turbulent models respectively. For Re3500, it was initially checked with the k-epsilon RANS model and later replaced by the k-omega SST model. A small study on different divScheme was also required at this point of the simulation. Then Re5000 and Re10000 models were analyzed under the k-omega SST RANS model. Similarly, for the turbulent flows in the rough pipe, Re5000 and Re10000 were modeled and analyzed using the k-omega SST model with the aid of nutKRoughWallFunction by replacing nutKWallFunction. This introduced new two parameters, Average roughness height, Ks, and roughness constant, Cs. This further opened up a way to carry out another study that connects the friction factor with the Ks value. The change in friction factor was studied through multiple simulations of the same case setup by only changing the Ks value.

All the numerical analysis results were noted, recorded, organized, and presented using tables and graphs. We mainly focused on velocity profile, maximum and minimum velocity, pressure/density value, and friction factor. Numerical results have been compared / validated against the analytical / empirical solutions available. This includes Reynold's equation, Darcy-Weisbach equation, Nikuradse's equations, Colebrook equation, Moody chart, and so on. This helped to understand how close are our numerical solutions. It also assisted us to question what went wrong at places where results are not a good match, thereby extending our study to another path and carrying out extra simulations and modifications.

Consider smooth pipe internal flows. Checking on the results of the velocity profile, the laminar flow, Re1000 gave a pretty good match. The plots and numerical values were exactly coinciding with the analytical results. This was the same with the transitional flow of Re2500, which was analyzed with the laminar model. Results were in pretty good agreement. The other transitional flow, Re3500 was simulated and validated using the turbulent model. Both k-epsilon and k-omega SST models were used for checking upon Re3500, but both weren't a good match. When a result comparative study was done for Re3500 using different divSchemes, the results were almost the same only. Turbulent flows, Re5000 and Re10000 showed a good match with Nikuradse's empirical power law velocity profile relation. Re10000 was a better match than Re5000. With the increase in Reynold's number, the profile got better. Considering the friction factor results flows simulated using the laminar model, Re1000 and Re2500 gave a pretty good report. For the transitional flow, Re3500, k-epsilon failed to give a result anywhere near the empirical value. The percentage error was over 100% which was not at all acceptable under any circumstances. Hence, we simulated the same case with the k-omega SST model, which in turn gave us a very good result. The result was pretty good, in which the percentage error was only around 3%. Checking on Re5000 and re10000 using k-omega SST models, the results were very satisfying with the empirical values. These are the observations noted for simulations with a smooth pipe, considering velocity profile and friction factor mainly. In a nutshell, we could say that everything worked fine for the smooth pipe. For every case, we were able to get pretty good matching results with laminar and k-omega SST RANS models in OpenFOAM.

Now, consider rough pipe internal flows. We were analyzing turbulent flows, Re5000 and Re10000. Velocity profile results were compared and validated against Nikuradse's empirical power law velocity profile and that was a good match with the increasing Reynold's number just like that in the smooth pipe. Coming to the friction factor, the results were not much different from that of the smooth pipes. The percentage error between the numerical results and empirical results was only around 10%, but it is still not satisfactory since the values didn't make much progress from that of the smooth pipes. Hence, the study was extended to average roughness height, Ks in nutKRoughWallFunction. Ks value was altered five times and simulations were run multiple times with the same case setup and analysis model. But, the friction factor didn't vary much according to the change in Ks. We were not able to pick up any specific trend or pattern between Ks and friction factor by altering Ks values. The result remained the same sometimes, while it abruptly changes some other times. These were the observations noted from the results of simulation rough pipe internal flows of Re5000 and Re10000, considering the velocity profile, friction factor, and roughness height.

Chapter 6

Conclusion

All the numerical results validated against analytical / empirical solutions and observations were noted, recorded, organized, and presented in tabular format or graphs. Results from the laminar model flows were perfectly agreeing with the analytical solutions in terms of velocity profile and friction factor. Re2500, the transitional flow was also successfully validated against the laminar model, which indicates that the flow is still under the laminar flow regime and not yet transformed into the transitional flow. But coming to Re3500 flow, we couldn't validate the velocity profile against the turbulent flow using k-epsilon or k-omega SST models. However, the k-omega SST model was successful in predicting the friction factor, whereas k-epsilon couldn't. Thus k-omega SST RANS model gave a pretty good result for transitional flow considering the friction factor. Considering the solution convergence, we simulated the Re3500 model multiple times with different divSchemes. Results received from those were pretty much the same only even though some converged and some didn't. The residual values came down to the order of nearly 10e-6 every case might be the reason for almost the same results for every divSchemes used. Results were in good agreement for the turbulent flows, Re5000 and Re10000. K-omega SST model offered matching results for both flows in terms of velocity profile and friction factor. And it was noted that the velocity profile is getting improved as Reynold's number increased. The profile became flattered at the center, the profile coincides better and numerical values came close to the empirical solutions.

When it comes to the rough pipe internal flows, numerical values didn't move much from that of the smooth pipes. The negligible roughness height value must be the reason behind this. When we did a comparative study between rough and smooth pipe values, the results are pretty much the same only. The percentage error still lies within 10% only, which is okay, but not so satisfactory. Hence another study on friction factor was conducted with varying values of Ks. Unfortunately, we couldn't observe any particular trend or pattern between the Ks value and friction factor. The values remain the same sometimes but change abruptly some other time.

Considering all the results we have, we could extend our studies in any different way. One of the possibilities is we could do the same analysis with different diameter pipes. The one we have used (0.00344m) is very small in dimension. Hence study can be further extended to larger diameter pipes and results can be noted. This would be very easy since we have used programmable blockMeshDict for creating geometry. So, by changing diameter and length directly in the blockMeshDict, we can create another geometry very fast. Another domain, where we can focus is transitional flows. Research and studies are going on in transitional flows for a very long time. So, we could extend the study in such a way we get a good match for the velocity profile. Coming to the rough pipe, more research is needed for getting better results for the friction factor. Our study cases didn't make much progress in results from that of the smooth pipes. We could also check whether larger diameter pipes / high Reynold's number flows could bring better results for the friction factor in rough pipes. Further studies on nutKRoughWallFunction and Ks value could reveal their relation with the frictional factor.

References

1. Muhammad Ahsan, Numerical analysis of friction factor for a fully developed turbulent flow using k–ε turbulence model with enhanced wall treatment, Beni-Suef University Journal of Basic and Applied Sciences, Volume 3, Issue 4, 2014, Pages 269-277, ISSN 2314-8535, https://doi.org/10.1016/j.bjbas.2014.12.001
2. Durst, F., Ray, S., Ünsal, B., and Bayoumi, O. A. (June 2, 2005). "The Development Lengths of Laminar Pipe and Channel Flows." ASME. J. Fluids Eng. November 2005; 127(6): 1154–1160. https://doi.org/10.1115/1.2063088
3. Salama, Amgad. 2021. "Velocity Profile Representation for Fully Developed Turbulent Flows in Pipes: A Modified Power Law" Fluids 6, no. 10: 369. https://doi.org/10.3390/fluids6100369
4. Fangqing Liu.: A Thorough Description Of How Wall Functions Are Implemented In OpenFOAM. In Proceedings of CFD with OpenSource Software, 2016, Edited by Nilsson. H., http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2016
5. Nering, Konrad, and Krzysztof Nering. 2021. "Validation of Modified Algebraic Model during Transitional Flow in HVAC Duct" Energies 14, no. 13: 3975. https://doi.org/10.3390/en14133975
6. Štigler, Jaroslav. (2012). Analytical Velocity Profile in Tube for Laminar and Turbulent Flow. https://doi.org/10.13140/2.1.3153.5046.
7. Turbulence modeling for CFD - by David C Wilcox - https://g.co/kgs/RQsFuq
8. Dzarma, Goziya & Adeyemi, Adekola & Taj-Liad, Abdulmalik. (2020). EFFECT OF INNER SURFACE ROUGHNESS ON PRESSURE DROP IN A SMALL DIAMETER PIPE. International Journal of Novel Research in Engineering & Pharmaceutical Sciences. 7. 1-8. https://www.researchgate.net/publication/341446473
9. Allen J.J, Shockling M.A, Kunkel G.J, and Smits A.J 2007, Turbulent flow in smooth and rough pipesPhil. Trans. R. Soc. A.365699–714, http://doi.org/10.1098/rsta.2006.1939
10. Babkin, V.A. Velocity profiles and heat exchange in turbulent flows through smooth and rough pipes. Dokl. Phys. 51, 605–609 (2006). https://doi.org/10.1134/S1028335806110085