DEVELOPED LAMINAR FLOW IN PIPE USING COMPUTATIONAL FLUID DYNAMICS

M. Sahu[1], Kishanjit Kumar Khatua[2] and Kanhu Charan Patra[3], T. Naik[4]

[1,2 &3] Department of Civil Engineering, National Institute of technology, Rourkela-769008,
[4]Padmanava college of engineering, Rourkela-769008, india

SYNOPSIS

In the analysis of water distribution networks, the design parameters are the lengths, diameters, and the coefficients friction of a pipe. Although some of these parameters such as the pipe lengths and the pipe diameters would remain the same at different points but the coefficients friction would change during the life of network and therefore they can be treated as imprecise information. A computational fluid dynamics (CFD) model of fully-developed laminar flow in a pipe is derived and implemented. This well known problem is used to introduce the basic concepts of CFD including, the finite-volume mesh, the discrete nature of the numerical solution, and the dependence of the result on the mesh refinement. The Numerical results are presented for a sequence of finer meshes, and the dependency of the truncation error on mesh size is verified experimentally. The comparison test results validate the analysis.

1. INTRODUCTION

The analysis of pipe flow is very important in engineering point of view. A lot of engineering problem dealt with it. Due to rigorous engineering application and implications the analysis is important. The flow of real fluid exhibits viscous effects in pipe flow. Here this effect is identified for laminar flow condition. The relationships defining fluid friction will be developed and will be defined to be applicable to laminar flow situations. The application of momentum equation is used to evaluate the friction loss coefficient. The expression defining the velocity distribution in a pipe flow across laminar flow is derived and demonstrated [1]. Hydro dynamically developed flow is achieved in a pipe after a certain length i.e. entrance length L_d, when the erect of viscosity reaches the center of the pipe. This is the point of concern of this experiment and by the help of CFD analysis package FLUENT, the problem is analyzed. After this point, the flow is essentially one-dimensional. Periodic boundary conditions are used when the physical geometry of interest and the expected pattern of the flow solution have a periodically repeating nature.

1Mrityunjay Sahu, B. Tech. Final year project student, Civil Engineering Students, NIT, Rourkela-769008, India, mrityunjay.nitrkl@gmail.com

2Asst. Professor, Department of Civil Engineering, NIT, Rourkela-769008, India, kkkhatua@yahoo.com

3 Professor, Department of Civil Engineering, NIT, Rourkela-769008, India, prof_kcpatra@yahoo.com

4. Trupti naik, faculty of padmanava college of engineering, Rourkela-769008,india, trupti_trn@yahoo.com

2. STEADY AND UNIFORM LAMINAR FLOW IN CIRCULAR CROSS-SECTION PIPES (GOVERINGING EQUATION)

The analysis of incompressible laminar flow will be done by the momentum equation of an element of flow in a conduit: the application of the shear stress-velocity relationship and knowledge of flow condition at the pipe wall which allows constant of integration to be evaluated.[3]

2.1 Assumptions

The flow acceleration is zero the resultant force on the element is zero for the steady flow condition.
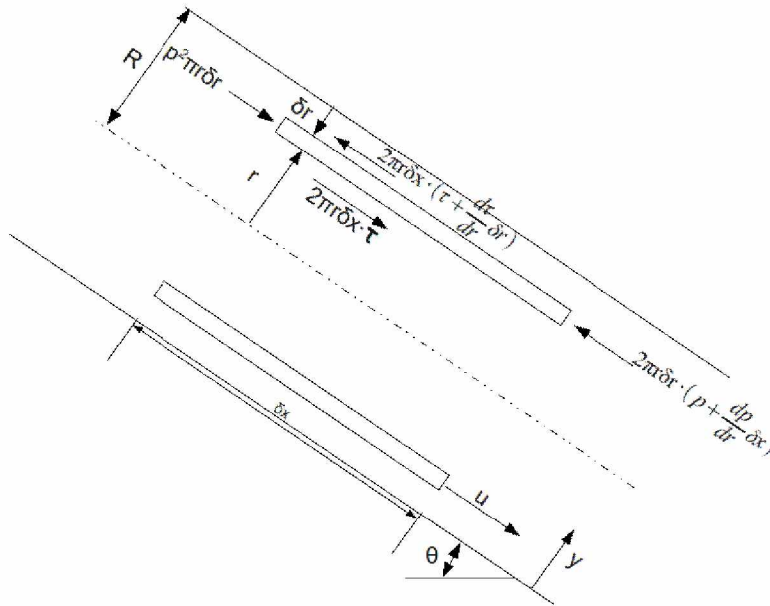

Fig. 1 Forces acting on annular element in a laminar pipe flow situation.

From Fig. 1.1 Applying change of momentum equation yields,

$$p2\pi r dr - (p + \frac{dp}{dx}dx)2\pi r dr + \tau \, 2\pi r dr - \left[2\pi r\tau\delta x + \frac{d}{dr}(2\pi r\tau dx)\delta r\right] + W\sin\theta = 0 \qquad (1)$$

Where $W = \rho g\delta x\delta r 2\pi r, \sin\theta = -\dfrac{dz}{dx}$

Now reducing the equation to:-

$$-\left(\frac{dp}{dx}\right) - \left[\frac{1}{dr}\frac{d}{dr}(r\tau)\right] - \rho g\left(\frac{dz}{dx}\right) = 0 \qquad (2)$$

By dividing δxδr2πr we get in above equation and rearranging;-

$$\frac{d}{dx}(p+\rho gz)+\frac{1}{r}\frac{d}{dr}(r\tau)=0 \tag{3}$$

Where (p + ρgz) is piezometric pressure is independent of r
By integrating above equation w.r.t r we get:-

$$\frac{r^2}{2}\left[\frac{d}{dx}(p+\rho gz)\right]+r\tau+\varepsilon=0 \tag{4}$$

The condition at pipe centerline are substituted in to above expression, then $\varepsilon = r = 0$
The shear stress-velocity gradient expression of above equation is employed in the direction of measurement of distance r from the center of the pipe, rather than the use of y measured from the pipe wall hence,

$$\tau=-\mu\frac{du}{dr} \tag{5}$$

Substituting for τ above equation (5) we get –

$$\frac{r^2}{2}\left[\frac{d}{dx}(p+\rho gz)\right]=r\mu\left(\frac{du}{dr}\right)-\varepsilon \tag{6}$$

And $\qquad du=\frac{r}{2\mu}\left[\frac{d}{dx}(p+\rho gz)+\frac{\varepsilon}{r\mu}\right]dr \tag{7}$

Integrating w.r.t. yields an expression for the velocity variation across the flow in terms of r and known system parameters:

$$u=-\frac{r^2}{4\mu}\left[\frac{d}{dx}(p+\rho gz)+\frac{\varepsilon}{\mu}\ln r+\varepsilon^{'}\right] \tag{8}$$

Values of ε, ἑ can be evaluated from boundary conditions at r = 0 and r = R. At r = R i.e. at the wall of the pipe, the local flow velocity u is zero hence:-

$$u=-\frac{R^2}{4\mu}\left[\frac{d}{dx}(p+\rho gz)\right] \tag{9}$$

And $\qquad u=-\frac{\left(R^2-r^2\right)}{4\mu}\left[\frac{d}{dx}(p+\rho gz)\right] \tag{10}$

This above equation describes the variation of local fluid u across the pipe and, from the form of the equation; the velocity profile can be seen to be parabolic. This equation also has a negative sign, due to fact that the pressure gradient will be negative in the flow direction.
The maximum velocity will occur on the pipe centerline, i.e. r = 0; hence,

$$u_{\max imum}=-\frac{\left(R^2-r^2\right)}{4\mu}\left[\frac{d}{dx}(p+\rho gz)\right] \tag{11}$$

Where

3. ENTRANCE LENGTH

Here the flow is internal flow which is constrained by boundary walls. The viscous effect will grow and meet also permeate the entire length of flow. In the pipe through the entrance region a nearly in viscid upstream flow converges and enters the tube. The viscous boundary layer grow downstream retarding the axial flow velocity u(r, x) at the wall and thereby accelerating the center-core flow to maintain the incompressible continuity requirement

$$Q = \int u.dA = \text{constant} \qquad (12)$$

At a finite distance from the entrance the boundary layer merges and in viscid or viscous flow disappears. The tube is entirely viscous and axial velocity adjust slightly further until at x = Ld. After this it no longer changes with varying x and is said to be fully developed .downstream the flow is almost constant and with constant shear .This length is generally the ENTRANCE LENGTH.

4. EXPERIMENTAL ANALYSIS

On flow of water in long , straight and uniform diameter pipes indicated that loss due to friction in a pipe in direct proportion with the velocity head, h_f and the distance between two sections, L and inversely proportional to pipe diameter D . Introducing a coefficient of proportionality f called the friction factor, Darcy and Weisbach proposed the following equation for head loss due to friction in pipe [4].

$$h_f = \frac{flv^2}{2gD} \qquad (13)$$

Making the dimensional, analysis of the problem of pipe friction, one finds that

$$f = F\left(R_e, \frac{k}{D}\right) \qquad (14)$$

Where $R_e = \dfrac{VD}{v}$,[4] the Reynolds's number

κ=mean height of roughness

$\dfrac{k}{D}$ = relative roughness.

Equation (2) suggests that the friction factor for pipes, having same Reynolds's number, roughness pattern and relative roughness will be same.

Basically the study of our flow having laminar flow condition. So for laminar flow condition i.e $R_e < 2000$,

$$f = \frac{64}{R_e} \qquad (15)$$

Hence the headless in laminar condition is independent of surface roughness.

h_f = velocity head,

R_e = Reynolds's number,

f = skin friction factor,

v = kinematic viscosity,

D = diameter of the pipe,

5. FLUENT:

Fluent is computational fluid dynamics (CFD) software package to stimulate fluid flow problems. It uses the finite volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, in viscid or viscous, laminar or turbulent etc.Geometry and grid generation is doing GAMBIT[7] which is the preprocessor bundled with FLUENT[6]

A Solution can be obtained by following these nine steps;

1. Create Geometry in GAMBIT.
2. Mesh Geometry in GAMBIT.
3. Set Boundary Types in GAMBIT.
4. Set Up Problem in FLUENT.
5. Solve!
6. Analyze Results.
7. Refine Mesh.

5.1 Modeling Details

The pipe is represented in 2D by a rectangle .The pipe geometry is displayed in figure
The procedure for solving the problem is;
- Create the geometry
- Set the material properties and boundary conditions
- Mesh the domain

Convergence limit is met or the number of iterations specified by the users is achieved and flow properties are set through parameterized case files. FLUENT converges the problem until the convergence limit is met or 100 number of iterations as specified is achieved.

5.2 Geometry

The geometry consists of a wall, a centerline, and periodic inlet and outlet boundaries. The radius and the length of the pipe can be specified.
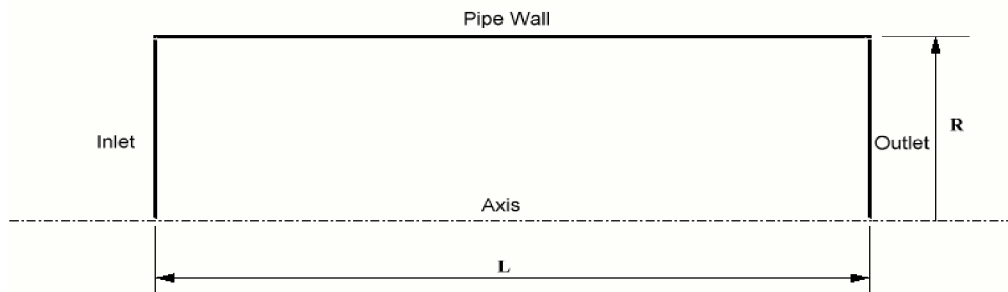

Fig 5.1 Schematic of the Flow Domain

5.3 Mesh

Coarse fine mesh types are available. Mesh density varies based upon the Refinement Factor.
The Refinement Factor values for the mesh densities are given in table 2.1

Table-5.3 Mesh Density Refinement Factor

Fine	1
Medium	1.414
Coarse	2

5.4 Refinement Factor

Using the Refinement Factor, First Cell Height is calculated with the following formula

$$FirstCellHeight = \frac{\text{Re}\, finemnetFactor \times YPlus \times (CharacteristicsLength \times Vis\cos ity)}{(0.199^{0.875} \times Velocity^{0.875} \times Density^{0.875})}$$ Reynolds

number is used to determine Y plus .Y plus values for turbulent flow conditions are summarized in table 5.3(A)

Table5.3 (A): Flow Regime vs. Reynolds Number

Reynolds Number Flow Regime Y plus/First Cell Height
Re = 2000 Laminar First Cell Height = Pipe Radius/38
2000<Re<15000 Turbulent, Enhanced Wall Treatment Y plus<5.0
Re = 15000 Turbulent ,Standard Wall Functions Y plus>30

The edges are meshed using the First Cell Height and the calculated number of intervals .The entire domain is meshed using a map scheme in figure 5.3(A

5.4 Physical models for FLUENT

Based on the Reynolds number, the following physical models are recommended;

Table 5.4: Turbulence Models Based on Pipe Reynolds Number

Re<2000 Laminar Flow
2000 = Re<10000 k- model
10000 = Re<15000 k-model
Re = 15000 k-model

If turbulence is selected in the Physics form of the operation menu, the appropriate turbulence model and wall treatment is applied based upon the Reynolds number.

5.5 Material Properties

The default material is air .The following material properties can be specified;
- density
- viscosity

Other materials such as Glycerin, Water and a User defined fluid can also be selected.

5.6 Boundary conditions

The mass flow rate or the fluid can be specified .The following boundary conditions are assigned in FLUENT[6];
Boundary assigned as

Table 5.6 boundary conditions

Inlet	Velocity inlet
Outlet	Pressure outlet
Centerline	Symmetry
Pipe Wall	Wall

5.7 SOLUTION

The mesh is exported to FLUENT along with the physical properties and the initial conditions specified. The material properties and the initial conditions are read through the case file .Instructions for the solvers are provided through a journal file. When the solution is converged or the specified number of iterations is met, FLUENT [6] exports data.

5.8 SCOPE AND LIMITATIONS

Transitional flow occurs between Reynolds numbers of <2000(laminar flow)[4].To improve accuracy of predictions, the κ -ε turbulence model was applied .However, a prediction error of five percent or more exists in the transitional region .Difficulty in obtaining convergence or poor accuracy may result if input values are used outside the upper and lower limits suggested in the problem overview.

5.9 Friction factor

In this soft-ware package friction factor [6] is calculated by taking density and velocity in to reference. Because the numerical differential equation depends on density as well as velocity. The friction factor equation used in it as

$$ f = \frac{\tau}{0.5\rho v^2} $$

Where ρ = density of the fluid;
V= average velocity of the fluid;

5.10 EXERCISE RESULTS

Report
XY Plots

The Plots reported by Flow Lab include:
- Axial velocity distribution along axis of pipe.
- Friction factor.

Available only when the flow is modeled as turbulent .An axial distribution of velocity along
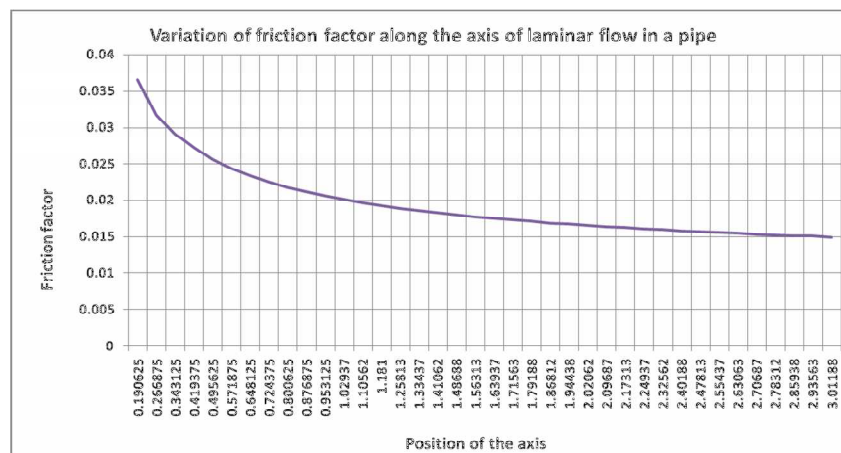

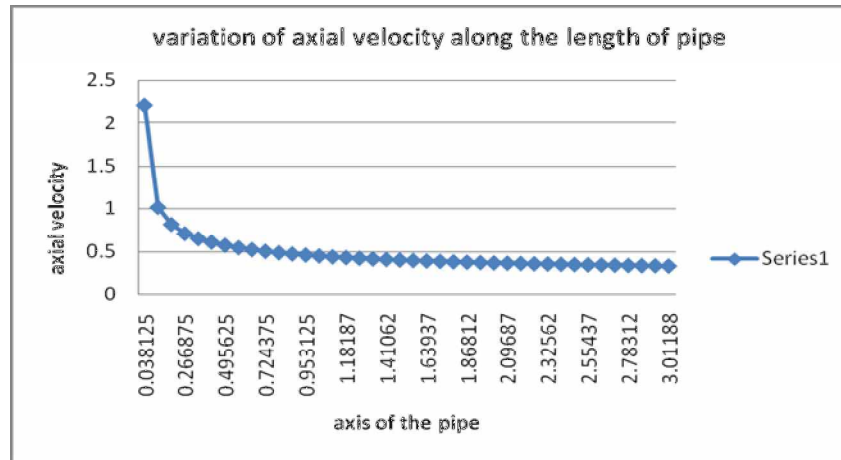**Fig. 5.10(A) friction factor V/S position of the axis**

Fig. 5.10(B) axial velocity v/s axis of the pipe

6. CONCLUSION

In this study the accuracy of numerical modeling of the laminar equation is investigated to determine the friction factor of pipe. The numerical differential equation is iterated and converged through the CFD package FLUENT[6] where the friction factor is found to be .0151 at the entrance length of 2.7068 m. the experimental result shows the friction factor is .0157 .The degree of accuracy of the numerical solution is checked with the experimental work done. This examination results indicates that the proposed model can be used with the highest degree of accuracy to determine the friction factor in comparison to the other methods.

Nomenclature :

P = the flow static pressure, R_e = Reynolds's number,

W = element weight , τ = shear stress along the wall,
τ = shear stress at distance r, ρ = density of liquid,

$\varepsilon, \acute{\varepsilon}$ = constant in equation   v = velocity of the flow,

ρ = density of fluid, f = skin friction factor,
μ = dynamic viscosity. κ = relative height of roughness,
D = diameter of pipe,

References

[1] White, Frank M.,"Viscous Fluid Flow", International Edition, McGraw-Hill, 1991.
[2] Adrian Bejan,"Convective Heat Transfer", John Wiley and Sons, 1994
[3] Stitching, H.,"Boundary-Layer Theory", 7thEdition.mcGraw-Hill, 1979.
[4] Douglas. Gasiorek, A.Swaffield,"Fluid Mechanics", PEARSON Education Asia, 2002
[5] White, Frank M,"Fluid Mechanics" International Edition, McGraw-Hill
[6] Fluent (Ansys corporation:) 6.3.26 user manual guide.
[7] Gambit 6.3.26 user manual guide.

Certificate

The author(s) certify that the paper titled "**DEVELOPED LAMINAR FLOW IN PIPE USING COMPUTATIONAL FLUID DYNAMICS**" and submitted for consideration for 7th International R&D Conference to be held in Bhubaneswar (India) from 4-6 February 2009, has not been published or presented at any other forum, in the proposed form.