International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 3, March - 2014

Analysis of Fully Developed Turbulent Flow in a AXI-Symmetric Pipe using ANSYS FLUENT Software

Manish Joshi[1], Priyanka Bisht[2], Dr. Anirudh Gupta[3]
[1] M. Tech Scholar, [2]M. Tech Scholar, [3] Associate Professor
Department of Mechanical Engineering,
Bipin Tripathi Kumaon Institute of Technology, Dwarahat, Almora, Uttarakhand (India) 263653

***Abstract*— This paper presents computational investigation of turbulent flow inside a pipe. In this paper, a axi-symmetric model of fully developed turbulent flow in a pipe is implemented with the help of ANSYS FLUENT 14.0 software and the variation of axial velocity and skin friction coefficient along the length of pipe is analysed. The fluids used for this purpose are Freon and ammonia. The results obtained computationally are found in well agreement with the results obtained analytically.**

***Keywords*— ANSYS FLUENT 14.0, Fully developed flow, Grid, Boundary layer, GAMBIT**

I. 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 turbulent flow condition. The application of momentum equation is used to evaluate the friction loss coefficient. The expression defining the velocity distribution in a pipe flow across turbulent flow is derived and demonstrated. 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.

The objective of the present work is to investigate the nature of fully developed turbulent flow in a pipe computationally and to determine the various parameters such as skin friction coefficient and centerline velocity associated with it.

II. MECHANISM OF INTERNAL FLOW

Although not all conduits used to transport fluid from one location to another are round in cross section, Most of the common ones are. These include typical water pipes, hydraulic hoses, and other conduits that are designed to withstand a considerable pressure differences across their walls without undue distortion of their shape. Typical conduits of noncircular cross section include heating and air conditioning ducts that are often of rectangular cross section. Normally the pressure difference between inside and outside of these duct is relatively small. Most of the basic principles involved are independent of cross section shape, although the details of flow may be dependent on it. Unless otherwise specified, we will assume that the conduit is round. We assume that the pipe is completely filled with the fluid being transported.

The fluid body is of finite dimensions and is confined by the channel or pipe walls. At the entry region to a channel, the fluid develops a boundary layer next to the channel walls, while the central "core" of the fluid may remain as a uniform flow. Within the boundary layer, viscous stresses are very prominent, slowing down the fluid due to its friction with the channel walls. This slowdown propagates away from the walls. As the fluid enters the channel the fluid particles immediately next to the walls are slowed down, these particles then viscously interact with and slow down those in the second layer from the wall, and so on. Downstream, the boundary layers therefore thicken and eventually come together, eliminating the central core. Eventually, the velocity assumes some average profile across the channel which is no longer

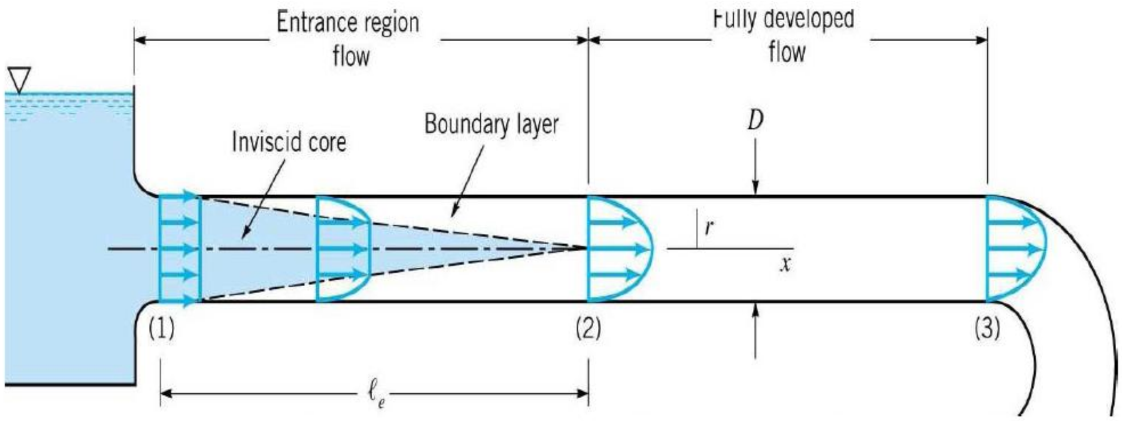

Figure1: Mechanism of internal flow

influenced by any edge effects arising from the entrance region. At this point, the flow no longer depends on what has occurred at the channel entrance, and we could solve for its properties (such as the velocity profile) without including an entrance region in the calculations. At this stage, we say that the flow has become "fully developed."

3. Literature review

A large number of research analyses have been carried out on the internal flows during the recent years. **Laufer, J.,** The structure of turbulence in fully developed pipe flow, NACA Report, NACA-TN-2954 (1953). **Powe and Townes [1973]** investigated the turbulence structure for fully developed flow in rough pipes. The method used to determine the turbulence structure involved examination of the fluctuating velocity spectra in all three coordinate directions. An important conclusion of this work was that in the central region of the pipe, the flow was relatively independent of the nature of the solid boundary.**Taylor (1984)** mathematically modelled the airflow through sampling pipes. Taylor (1984) begins by stating that for a steady incompressible fluid flow through a smooth pipe, the energy conservation equation can be used. He quoted Darcy's formula forehead loss in pipes caused by friction. He also commented that this equation is applicable to either laminar or turbulent flow. In more recent work, **Koh [1992]** presented an equation to represent the mean velocity distribution across the inner layer of a turbulent boundary layer, and used this velocity profile to derive a friction factor correlation for fully developed turbulent pipe flow. **Cole (1999)** investigated the disturbances to pipe flow regimes by jet induction to improve the available techniques to mathematically model the performance of aspirated

smoke detection systems. He stated that there is a significant area of uncertainty in determining the friction factor and it has not been established that the friction factor is unaffected by upstream disturbances to the flow regime whether that regime is turbulent, laminar or transitional. He suggested that the assumption that the flow regime can be regarded as fully developed may not be true. Similar to the work carried out by Taylor (1984), Cole (1999) suggested that the energy losses in any pipe fitting can be broken down into three components: entry loss, exit loss and friction losses. **Saho et al. (2009)** investigated the accuracy of numerical modelling of the laminar equation to determine the friction factor of pipe. The numerical differential equation is iterated and converged through the CFD package FLUENT where the friction factor is found to be 0.0151 at the entrance length of 2.7068 m. while the experimental result shows the value of friction factor as 0.0157.Besides these previous works, a number of formulations and analytical results have been discussed in various books. The expression defining the velocity distribution in a pipe flow across turbulent flow is derived and demonstrated in **Bejan, "Convective heat transfer coefficient",1994** . Hydro dynamically developed flow is achieved in a pipe after a certain length i.e. entrance length *Le* , where the effect of viscosity reaches the centre of pipe. At this point the velocity assumes some average profile across the pipe which is no longer influenced by any edge effects arising from the entrance region. The flow of real fluids exhibit viscous effects in pipe flow. Here this effect is identified for turbulent flow conditions. The relationships defining friction in pipes have been demonstrated in **White, F.M., Fluid Mechanics, 3rd edition, 1994**. 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 demonstrated in **Stitching, H., "Boundary-Layer Theory", 7thEdition.mcGraw-Hill, 1979.**. 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 in **White, Frank M.,"Viscous Fluid Flow", International Edition, McGraw-Hill, 1991**.

Nomenclature

Vc	Centerline Velocity or Axial Velocity m/s	*R*	Radius of elementary ring, m
R	Radius of Pipe, m	*τw*	Wall shear stress
D	Diameter of Pipe, m	*Δp*	Change in pressure, N/m2
L	Length of Pipe, m	*V*	Velocity of flow, m/s
Q	Volume flow rate, m3/s	*ρ*	Density of fluid, kg/m3
Cf	Skin friction coefficient	*f*	Friction factor
n	Function of the Reynolds number		

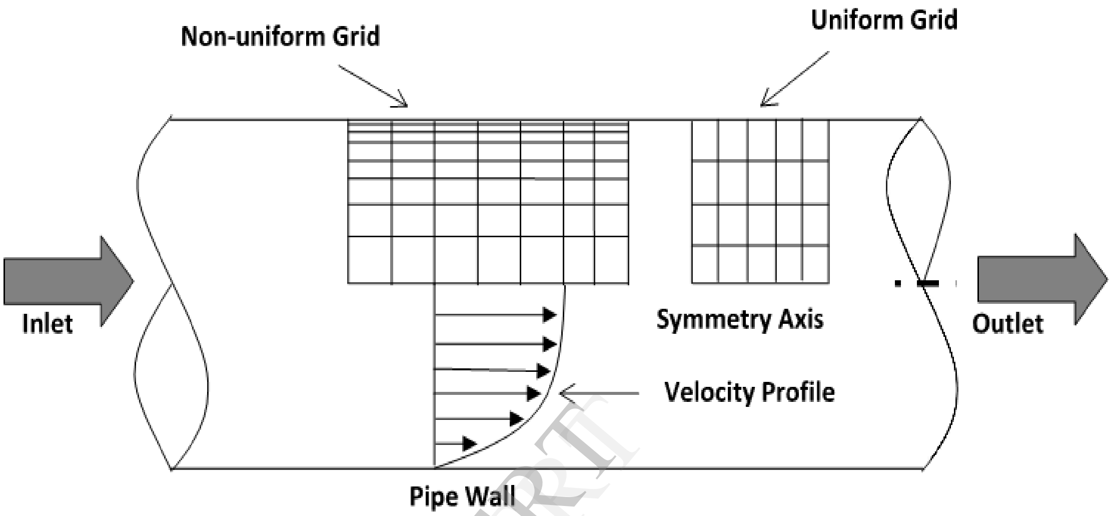

Figure 2: Pipe Geometry

III. Analytical Solution

The correlation for the velocity profile in turbulent flow is given by

$$\frac{u}{Vc} = \left(1 - \frac{r}{R}\right)^{\frac{1}{n}}$$

Where *u* is the time mean average of x- component of instantaneous velocity, V_c is the centreline velocity or axial velocity, *R* is the radius of pipe, *r* is the radius of elementary ring and *n* is a function of the Reynolds number. To determine the centreline velocity, V_c, we must know the relationship between *V* (the average velocity) and V_c. . This can be obtained by integration of equation (1). Since the flow is axisymetric,

$$Q = AV = \int u dA = V_c \int_{r=0}^{r=R} (1-\tfrac{r}{R})^{\frac{1}{n}} (2r)\, dr$$

$$Q = 2\pi R^2 V_c \frac{2n^2}{(n+1)(2n+1)}$$

Since $Q = \pi R^2 V$, therefore we get

$$\frac{V}{V_c} = \frac{2n^2}{(n+1)(2n+1)}$$

The formula for calculating the value of skin friction coefficient is given by

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho v^2}$$

Where,

τ_w is the wall shear stress is given by

$$\Delta_p = \frac{L}{D}\frac{\rho V^2}{2}$$

Where f is the friction factor and is calculated with the help of Moody chart.

Input Parameters

S.No.	Parameter	Turbulent flow
1	Diameter of pipe (m)	0.2
2	Length of pipe(m)	40
3	Flowing Fluid	Freon, Ammonia
4	Temperature (K)	293,293
5	Density of fluid (Kg/m^3)	1330,612
6	Viscosity of fluid (Kg/ms)	$2.6334e^{-04}$, $2.19096e^{-04}$
7	Velocity of fluid at inlet (m/s)	0.01,0.005
8	Outside Pressure (atm)	1,1
9	Flow Model	K-ε model
10	Material of pipe	copper

5. Modelling and Simulation

The whole analysis is carried out with the help of software "ANSYS Fluent 14.0". ANSYS Fluent 14.0 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 Geometry and grid generation is done using GAMBIT which is the pre-processor bundled with FLUENT. The two dimensional computational domain modelled using hex mesh for models are as shown in fig 2. The complete domain of axi-symmetric tube

consists of 21636 nodes 21000 Elements. Grid independence test was performed to check the validity of the quality of the mesh on the solution. Further refinement did not change the result by more than 0.9% which is taken as the appropriate mesh quality for computation.

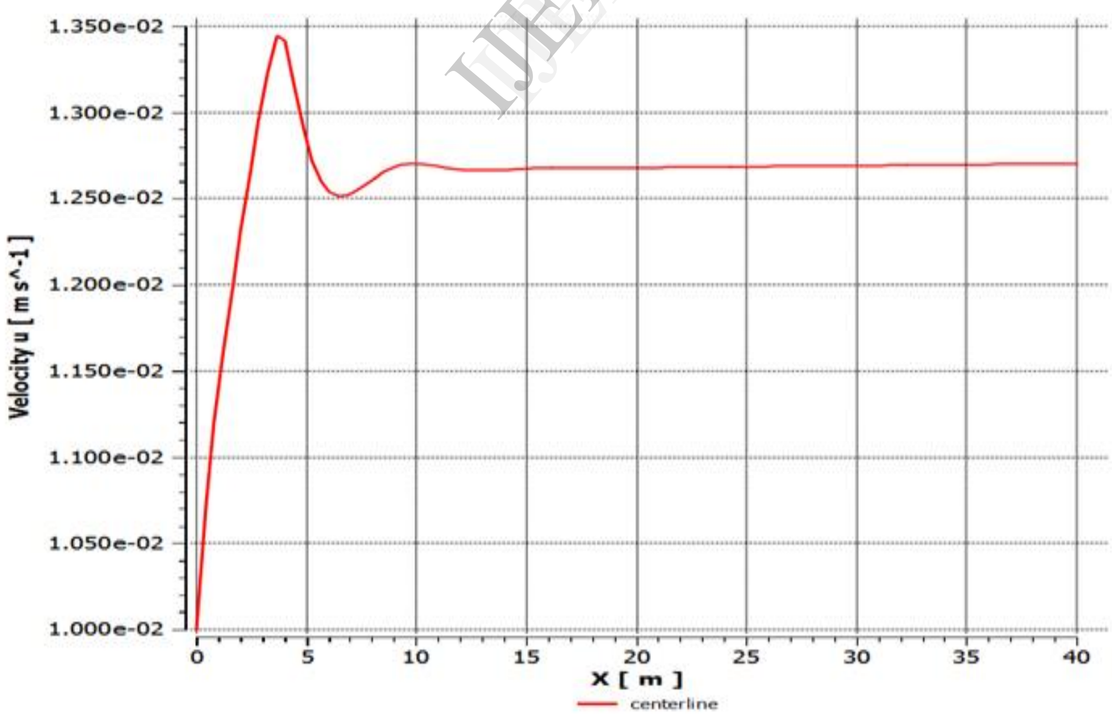

Figure 3: Axial Velocity of Freon along the position of pipe

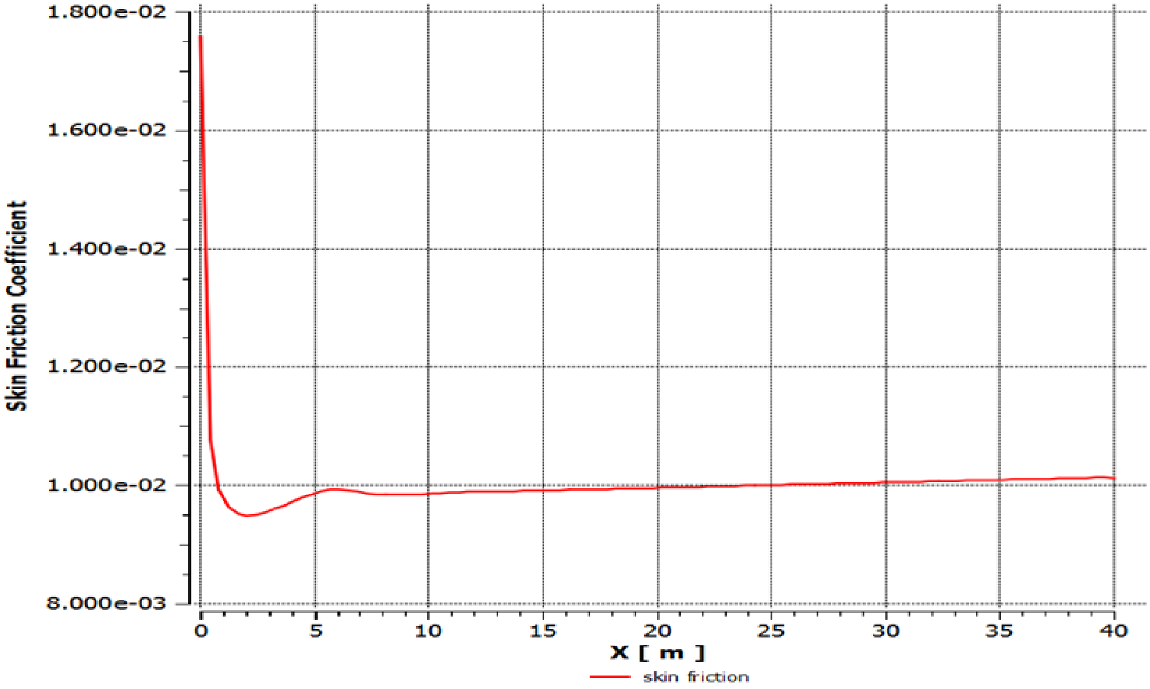

Figure 4: Skin friction coefficient of Freon along the position of pipe

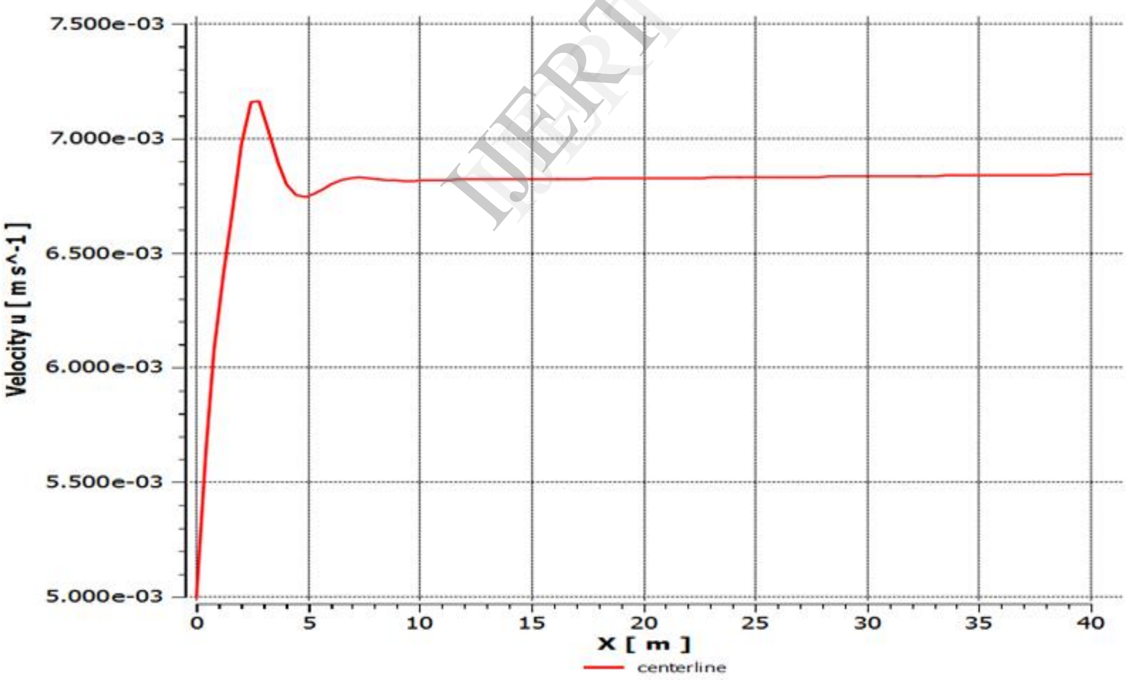

Figure 5: Axial velocity of ammonia along the position of pipe

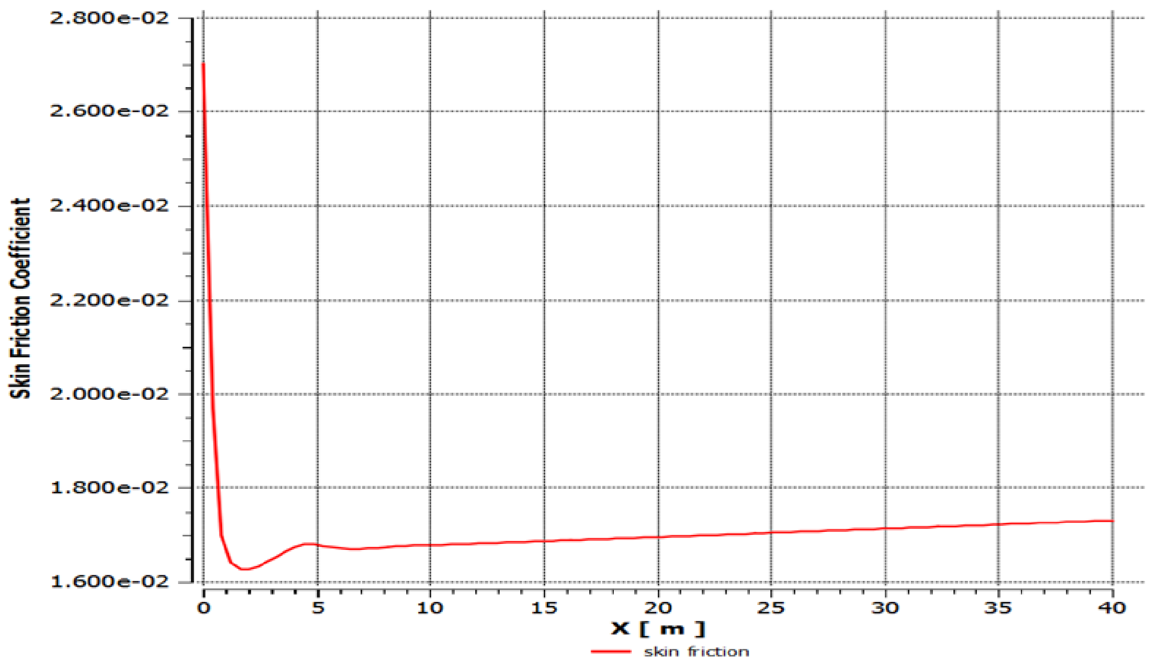

Figure 6: Skin friction coefficient of ammonia along the position of pipe

IV. Results and Discussion

For turbulent case of Freon as shown in figure 3, the centreline velocity for fully developed region is around 0.012m/s while the value calculated analytically is 0.0127m/s. Similarly, for turbulent case of Ammonia, the value of centreline velocity for fully developed region according to figure 4 is 0.00679m/s while the value obtained analytically is equal to 0.0068m/s. Similarly, for fully developed turbulent flow of Freon and Ammonia, the value of skin friction coefficient comes out to be 0.01075and 0.01075 respectively while the values obtained computationally are 0.0100 and 0.01650 (figure 5 and figure 6). It is also observed from the results that the axial velocity against position of centerline also reveal that the axial velocity increases along the length of pipe and after some distance it becomes constant which is in conformity to the results obtained experimentally. The results of the skin friction coefficient against position of centerline also reveal that the skin friction decreases along the length of pipe and after some distance it becomes constant which is in conformity to the results obtained experimentally

V. Conclusion

Based on the CFD analysis of the flow inside the pipe the following conclusions can be drawn:

1. Computed friction factors and axial velocity were found in close agreement with the analytical values.
2. Skin friction coefficient decreases along with the length of pipe and becomes constant after entering the fully developed regime.
3. Axial velocity increases along with the length of pipe and in the fully developed regime it becomes constant.
4. CFD analysis represents successfully the hydrodynamic of the system.

References

[1] LAUFER, J., The structure of turbulence in fully developed pipe flow, NACA Report,NACA-TN-2954 (1953).

[2] Powe, R. E. and Townes, H. W., \Turbulence structure for fully developed flow in rough pipes," J. Fluids Eng., pp. 255-261, 1973.

[3] Taylor, N.A. (1984) Modeling of Air Flows Through the Sample Pipe of a Smoke Detecting System, United Kingdom Atomic Energy Authority
Koh, Y., Turbulent flow near a rough wall," J. Fluids Eng., Vol. 114, pp. 537{542, 1992.

[4] Cole, M., Disturbance of Flow Regimes by Jet Induction, PH.D Thesis, Victoria University of Technology, Australia, 1999.

[5] M. Sahu, Kishanjit Kumar Khatua and Kanhu Charan Patra, T. Naik, Developed laminar flow in pipe using computational fluid dynamics, 2009,7th International R & D Conference on Development and Management of Water and Energy Resources, 4-6 February2009, Bhubaneswar, India

[6] Adrian Bejan,"Convective Heat Transfer", John Wiley and Sons, 1994.

[7] White, F.M., Fluid Mechanics, 3rd edition, 1994.

[8] Schlichting, H.,"Boundary-Layer Theory", 8th Edition. mcGraw-Hill, 2001.

[9] White, Frank M.,"Viscous Fluid Flow", International Edition, McGraw-Hill, 1991.