International Journal of Engineering Inventions
e-ISSN: 2278-7461, p-ISSN: 2319-6491
Volume 3, Issue 12 [December. 2014] PP: 77-81

Practical Analysis Of Turbulent Flow In A Pipe Using Computational Fluid Dynamics

V.RajaKiranKumar[1], Dr.H.N.Suresh Kumar[2], Ch.Kiran Kumar[3]
[1] (Associate Professor, Mechanical Engineering Department, CMR Engineering College, Hyderabad)
[2](Professor, Mechanical Engineering Department, CMR Engineering College, Hyderabad).
[3](Assistant Professor, Mechanical Engineering Department, CMR Engineering College, Hyderabad.)

Abstract: This paper presents computational investigation of turbulent flow inside a pipe. In this paper, a computational fluid dynamics (CFD) model of fully developed turbulent flow in a pipe is implemented with the help of ANSYS FLUENT 12.0 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 air and water. The results obtained computationally are found in well agreement with the results obtained analytically.

Keywords: ANSYS FLUENT 12.0, Fully developed flow, Grid, Boundary layer, GAMBIT

I. INTRODUCTION

The transport of fluid in a closed conduit is extremely important in our daily operations. A brief consideration of the world around us will indicate that there is a wide variety of applications of pipe flow. Such applications range from the large, man-made Alaskan pipeline that carries crude oil almost 1290 km across Alaska, to the more complex natural systems of “pipes” that carry blood throughout our body and air into and out of our lungs. Other examples include the water pipes in our homes and the distribution system that delivers the water from the city well to the house. The analysis of pipe flow is also very important in engineering point of view. A lot of engineering problem dealt with it. Due to rigorous engineering application and implications, it has become quite necessary to carry out research analysis on the nature of flow inside pipes and tubes. 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 centreline velocity associated with it.

II. MECHANISM OF INTERNAL FLOW

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.

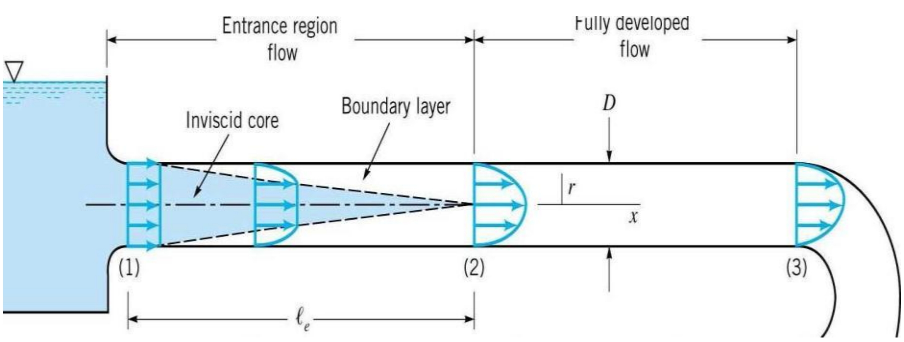

Fig1: Mechanism of internal flow

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 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."

III. LITERATURE REVIEW

A large number of research analyses have been carried out on the internal flows during the recent years. **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. **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 L_e , 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**.

Nomenclature		V	Velocity of flow, m/s
V_c	Centreline Velocity or Axial Velocity,m/s	ρ	Density of fluid, kg/m^3
R	Radius of Pipe, m	f	Friction factor
D	Diameter of Pipe, m	C_f	Skin friction coefficient
L	Length of Pipe, m	n	Function of the Reynolds number

IV. ANALYTICAL SOLUTION

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

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

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,

$$Q = AV = \int u\mathrm{d}A = V_c \int_{r=0}^{r=R} \left(1 - \frac{r}{R}\right)^{1/n} (2\pi r)\, \mathrm{d}r \qquad (2)$$

$$Q = 2\pi R^2 V_c \frac{n^2}{(n+1)(2n+1)} \qquad (3)$$

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

$$\frac{\quad}{V_c} = \frac{2n^2}{(n+1)(2n+1)} \qquad (4)$$

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

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho V^2} \qquad (5)$$

Where,

τ_w is the wall shear stress and is given by

ΔpD

$$\tau_w = \frac{}{4L} \qquad (6)$$

The pressure drop Δp is given by

$$\Delta p = f\frac{L}{D}\frac{\rho V^2}{2} \qquad (7)$$

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

V. MODELLING AND SIMULATION

The whole analysis is carried out with the help of software „ANSYS Fluent 12.0‟. ANSYS Fluent 12.0 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.

VI. RESULTS AND DISCUSSION

For turbulent case of air as shown in fig 2 , the centreline velocity for fully developed region is around 1.19m/s while the value calculated analytically is 1.22m/s. Similarly, for turbulent case of water, the value of centreline velocity for fully developed region according to figure 3 is 0.061m/s while the value obtained analytically is equal to 0.06122m/s.

Table 1.Input Parameters

S.No.	Parameter	Turbulent flow
1	Diameter of pipe (m)	0.2
2	Length of pipe(m)	8
3	Flowing Fluid	Air, Water
4	Temperature (K)	283 , 283
5	Density of fluid (Kg/m^3)	1.240722, 1000
6	Viscosity of fluid (Kg/ms)	176.10e-07, 1.3038e-03
7	Velocity of fluid at inlet (m/s)	1 , 0.05
8	Outside Pressure (atm)	1,1
9	Flow Model	K-ε model
10	Material of pipe	Steel

Similarly, for fully developed turbulent flow of air and water, the value of skin friction coefficient comes out to be 0.01and 0.009 respectively while the values obtained computationally are 0.00795 and 0.01 (figure 4 and figure 5).

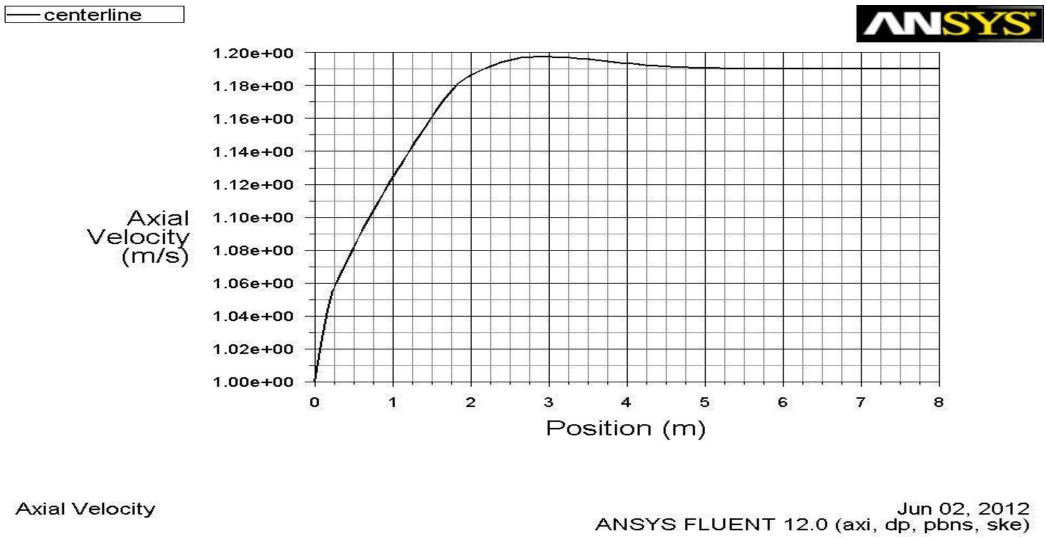

Fig 2: Axial Velocity of air along the position of pipe

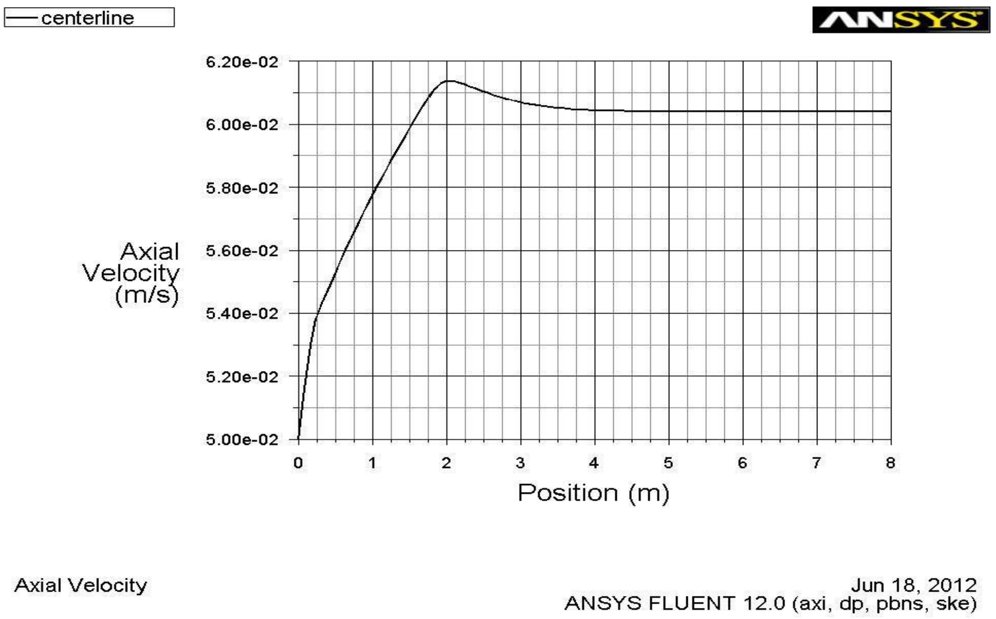

Fig 3: Axial velocity of water along the position of pipe

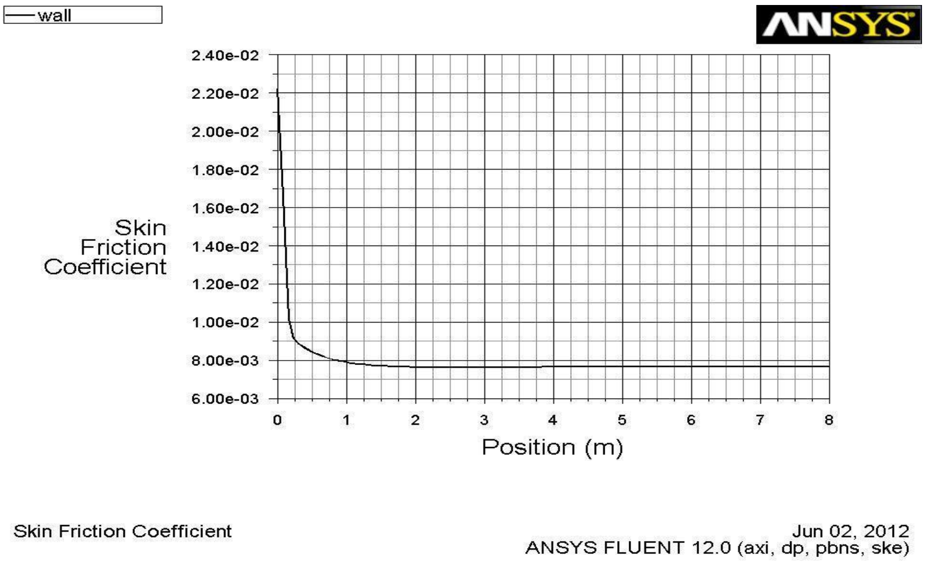

Fig 4: Skin friction coefficient of air along the position of pipe

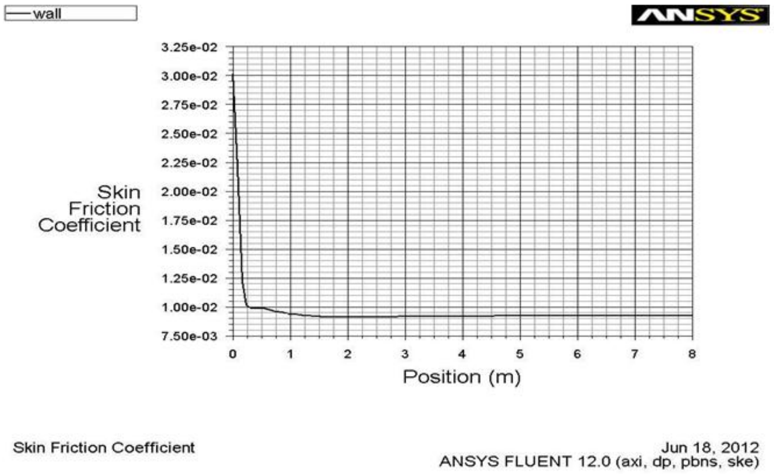

Fig 5: Skin friction coefficient of water along the position of pipe

It is also observed from the results that the axial velocity against position of centreline 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 centreline 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.

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

[2]. Benedict, R. Fundamentals of Pipe Flow, 3rd edition. A. Wiley – Inter Sciences, 1980.

[3]. Taylor, N.A. (1984) Modeling of Air Flows Through the Sample Pipe of a Smoke Detecting System, United Kingdom Atomic Energy Authority

[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]. White, F.M., Fluid Mechanics, 3rd edition. Mc-Graw Hill, 1994.