Application of CFD analysis in fully developed Velocity and Temperature flow field through pipe

[a] Siddhesh Tirodkar, [b]Siddappa Bhusnoor
[a]M.Tech student , [b] Associate professor,
Department of Mechanical Engineering, K. J. Somaiya College of Engineering, Vidyavihar-400077
[a]siddhesh.t@somaiya.edu

Abstract: Flow through pipe is observed at many places in everyday life. To understand the effect of velocity and temperature profile on such flows, many researchers performed experiments and estimated empirical relations which explain the flow effectively. In the present work primary focus is on the viability of results obtained from CFD analysis and compared those with numerical calculations. It is observed that assumptions considered during formulation need to be followed carefully. The realizable k-ε turbulent model found more appropriate than standard k-ε turbulence model for flow through pipe.

Key Words*:* Computational fluid dynamics, Velocity profile, Temperature profile

1. INTRODUCTION

Flow of fluid in the close ducts has many common applications in our daily life for example Blood flow in veins, waste management system of living organisms, water pipelines in building, pipe lines carrying gas and oil from Deep Ocean to land, engine exhaust from engine, etc. Many engineering problems are associated with this because of its large spread. The behavior of fluid flowing through such close duct and its effect on velocity profile, friction factor and temperature profile helps to understand flow characteristics. Fluid flow is divided into laminar or turbulent flow depending on the magnitude of Reynolds number. The Reynolds number (Re) is the ratio of momentum forces to viscous forces and consequently quantifies the relative importance of these two types of forces for the given flow conditions. Laminar flow is a flow regime characterized by high momentum diffusion and low momentum convection. Turbulent flow is a flow regime characterized by chaotic property changes. This includes low momentum diffusion, high momentum convection, and rapid variation of pressure and flow velocity in space and time. For flow in a pipe, experimental observation shows that for "fully developed" flow, laminar flow occurs when $Re < 2300$ and turbulent flow occurs when $Re > 4000$ Reynold [8]. In between these two regions the flow is in transition mode where it shows property of both regions.

Pipe flow is a kind of internal flow, which is mainly consists of two regions, first developing region or entry length region where the flow is inconsistent along length the pipe and other is fully developed region. According to fluid mechanics, in hydrodynamic entry length region the boundary layer develops along the pipe, at core, the velocity is high compared to wall due to presence of viscous forces near the vicinity of wall. After certain length the boundary layer meets at center, and the flow becomes consistent. Second region is developed region; or fully developed flow , which implies that implies that the velocity profile does not change in the fluid flow direction hence the momentum also does not change in the flow direction. In such a case, the pressure in the flow direction will balance the shear stress near the wall. Fig. 1 describes internal flow mechanics of flow through pipe for velocity and temperature field. So one can say that entry length plays vital role in internal pipe flows where edge effect is significant. Also, similar kind of thermal entry length region is observed in the temperature profile in pipe. Computational fluid dynamics, usually abbreviated as CFD, which is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze the problem that involves fluid flows is used from past few decades to understand fluid behavior in many applications.

Many researchers have analyzed the internal flows through pipe in recent years. Mathematical modeling of airflow through sampling pipes is done by Taylor [9] . Sahu [6] investigated the accuracy of numerical modeling of the laminar equation for friction factor of pipe along with evaluation of the effect of mesh refinement factor. Basic concepts of flow through pipe are explained in detailed by Kumar [5], Cengel [1], White [11] for fully developed flow in close ducts for laminar and turbulent conditions. Computational fluid dynamics analysis and its basic principles are well explained in Versteeg [10]. No significant CFD and numerical comparison work for temperature profile in a pipe with computational and numerical calculation for constant wall temperature is observed in literature.

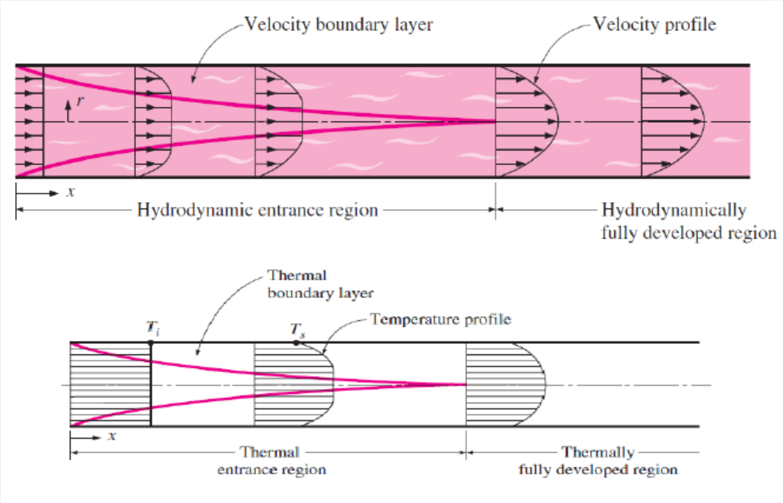

Figure 1. Internal flow mechanics for velocity and temperature profile (Source: Cengel [1])

2. METHODOLOGY

2.1 Numerical calculation

2.1.1 Velocity calculation

Hydrodynamic entry length region is part of pipe where the flow is inconsistent along the length of pipe. Laminar flow is calculated using following equation as suggested by Kays [4], Bhatti [2],

$$\frac{L}{D} = 0.05\ Re \tag{1}$$

Where, L= entry length distance (m); D = diameter of pipe (m); Re = Reynolds number

Turbulent flow is calculated using following equation, explained by Bhatti [2], Zhi-qing [12]

$$\frac{L}{D} = 1.359\ Re^{0.25} \tag{2}$$

To find centerline velocity or axial velocity, it is important to know the relation between average velocity and centerline velocity. Laminar flow is derived based on Cengel [1]

$$u(r) = 2V(1 - \frac{r^2}{R^2}) \tag{3}$$

Where, u = time average x velocity component (m/s), r = radius of elementary ring (m), V = average velocity (m/s)

For axial velocity r = 0 hence,

$$Vc = 2 * V \tag{4}$$

Where, Vc= centreline velocity or axial velocity (m/s)

Turbulent flow is derived based on Cengel [1]

$$\frac{u}{V_c} = (1 - \frac{r}{R})^{\frac{1}{n}} \tag{5}$$

Where, n is a function of the Reynolds number.

To find centerline velocity, calculating volume flow rate is calculated as,

$$Q = AV = \int_{r=0}^{r=R} \left(1 - \frac{r}{R}\right)^{\frac{1}{n}} (2\pi r)dr = 2\pi R^2 V_c \frac{n^2}{(n+1)(2n+1)} \qquad (6)$$

By comparing the equation for turbulent flow condition centerline velocity becomes

$$Vc = V * \frac{(2n+1)(n+1)}{(2n^2)} \qquad (7)$$

Following equation is used for calculating the value of skin friction coefficient,

$$C_f = f/4 \qquad (8)$$

Where, C_f =Skin friction coefficient , f =friction factor

It is observed experimentally that for laminar flow $f = \frac{64}{Re}$, and for turbulent flow $f = 0.316\ Re^{-0.25}$

2.1.2 Temperature calculation

Thermal entry length suggested by Kays [4] for laminar flow is as follows ,

$$\left(\frac{X}{D}\right) \cong 0.05\ Re\ Pr \qquad (9)$$

Where, X=Thermal entry length (m), D=pipe diameter (m), Re=Reynold number Pr=Prandtl number and for turbulent flow is as follows,

$$\left(\frac{X}{D}\right) = 10 \qquad (10)$$

For constant surface temperature, change in temperature is as follows.

$$\frac{T_s - T_{m(x)}}{T_s - T_{m,i}} = \exp\left(-\frac{Px}{mC_p}\bar{h}\right) \qquad (11)$$

Where, T_s = surface temperature (m), $T_{m(x)}$ = mean temperature at x distance (K), $T_{m,i}$ = mean inlet temperature (K), P = pipe periphery (m), x = distance from 0 to L, m = mass flow rate (kg/s), Cp = specific heat capacity (J/kgK), $\bar{h}$=average heat transfer coefficient.(W/m^2K)

(*Note that velocity and temperature profile calculations are done after flow develops completely, i.e., in fully developed region).

From Nusselt number we can say that,

$$h = \frac{NuK}{D} \qquad (12)$$

For laminar flow, value of Nusselt number is 3.66 Kays [4].

For turbulent flow, value of Nusselt number given by Gnielinski [3]

$$Nu = \frac{(f/8)(Re-1000)Pr}{1+12.7(f/8)^{1/2}(Pr^{2/3}-1)} \qquad (13)$$

Where, Nu = Nusselt number, f = friction factor, Re = Reynold number, Pr = Prandtl number

Value of f is given by Petukhov [7] is

$$f = (0.790\ln Re - 1.64)^{-2} \qquad (\text{For } 3000 \leq Re \leq 5E06) \qquad (14)$$

Where, f = friction factor, Re = Reynold number

2.2 Simulation

Ansys fluent 14.0 has been utilized to simulate the flow through pipe. It uses the finite volume method to solve the governing equations for a fluid Geometry (Fig. 2). We have used 2D geometry to simulate the data. The mesh has been optimized by monitoring pressure difference between inlet and outlet (Fig. 3). Pressure based solver with steady state conditions are used to simulate the laminar and turbulent flow. Particularly for turbulent flow, k-ε model with standard wall treatment is set during simulation. Standard atmospheric temperature is considered around the pipe and in the flow stream. Energy model is activated to compute the temperature changes in flow field. Matlab 2014a software has been used for data analysis. Table 1 represents the parameters utilized while simulating flow through pipe for different Reynolds number.

TABLE 1. PARAMETER USED FOR COMPUTATION

Sr. No	Parameter	Value
1	Diameter of pipe (m)	0.015
2	Length of pipe (m)	1
3	Flowing fluid	Air
4	Temperature (k)	443
5	Viscosity of fluid (kg/ms)	2.478E-05
6	Density of fluid (kg/m3)	0.787594
7	Reynolds number	100, 500, 1000, 1500, 2000, 4000, 5000,6000,7000,8000,10000,12000
8	Outside pressure (atm)	1

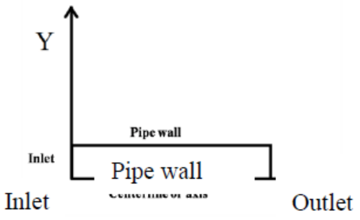

Figure 2: Centerline or axis for simulation

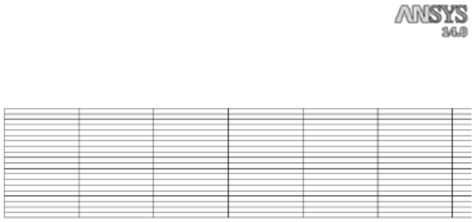

Figure 3: Mesh structure of pipe used for simulation

3. RESULTS AND DISCUSSIONS

The values obtained from equations and outputs of , it is found that all the simulations carried out in Ansys are in sync with the theories indexed in literature. Skin friction coefficient for different Reynolds number along the length of pipe in laminar and turbulent flow are shown in Fig 4 and Fig. 5. It is found that with increasing value of Reynolds number the value of Skin friction coefficient decreases gradually in laminar flow, but in turbulent flow the difference is found to be smaller between two Reynolds number . Axial velocity along length of pipe for different Reynolds number for laminar and turbulent flow conditions (Fig. 6 & Fig. 7). With increasing Reynolds number, the hydrodynamic entry length region increases in laminar flow, whereas in turbulent flow the hydrodynamic entry length observed to be very small. Skin friction coefficient is calculated at the end of pipe for laminar and turbulent flow (Fig. 8 & Fig. 9) . It is observed that for laminar flow both the equation value and simulated results are in well agreement, but in turbulent flow the values obtained from equation and simulated results shows some deviation. For turbulent flow simulation in Ansys, the results obtained from standard k-ε turbulent model are close to numerically calculated data.

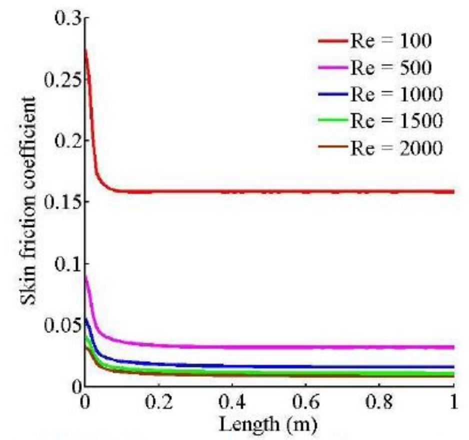

Figure 4: Skin friction coefficient for laminar flow

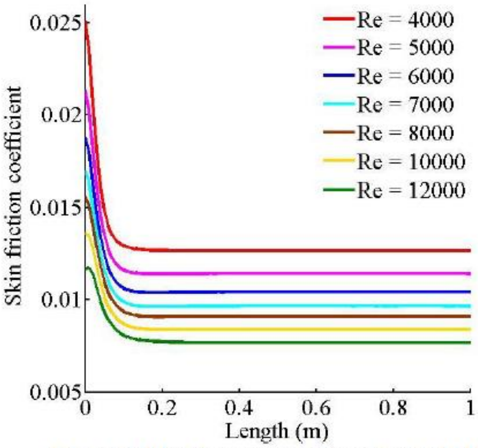

Figure 5: Skin friction coefficient for turbulent flow

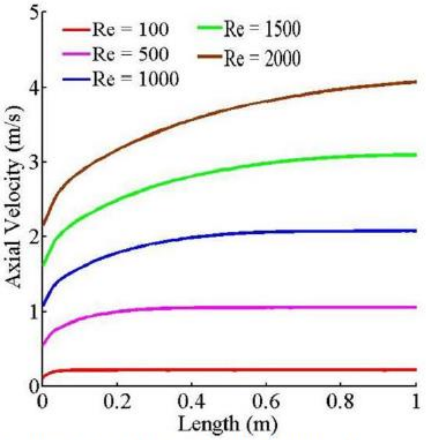

Figure 6: Axial velocity for laminar flow

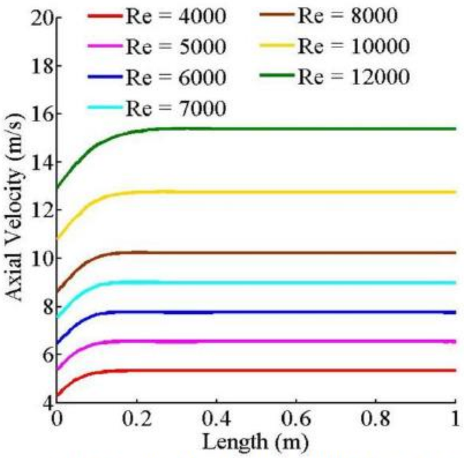

Figure 7: Axial velocity for turbulent flow

Axial velocity at the end of pipe for laminar and turbulent flow (Fig. 10 & Fig. 11). Similar trends like that of skin friction coefficient is obtained in mean axial velocity plots. It is observed that realizable k-ε turbulent model is close to the equation values. Mean exit temperature at the end of pipe for laminar and turbulent flow (Fig. 12 & Fig. 13) with constant wall temperature case. It is observed that in laminar flow the deviation is constant for higher Reynolds number. In turbulent flow case again the realizable k-ε turbulent model found to be more suitable to use.

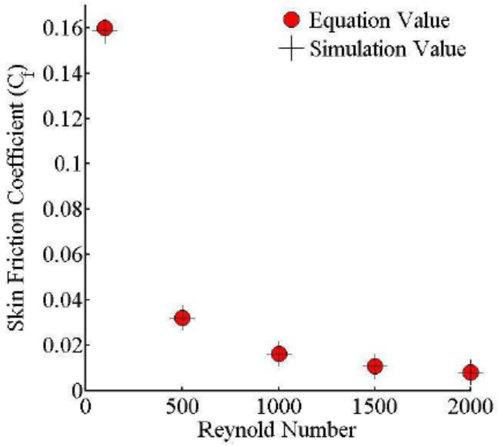

Figure 8: Skin friction coefficient comparison for laminar flow (at pipe end)

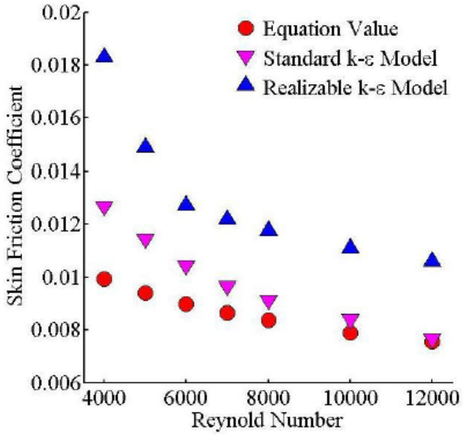

Figure 9: Skin friction coefficient comparison for turbulent flow (at pipe end)

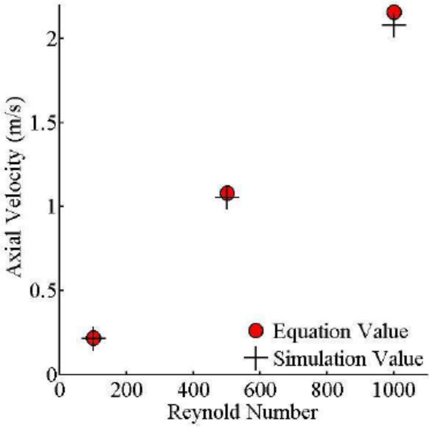

Figure 10: Axial velocity comparison for laminar flow (at pipe end)

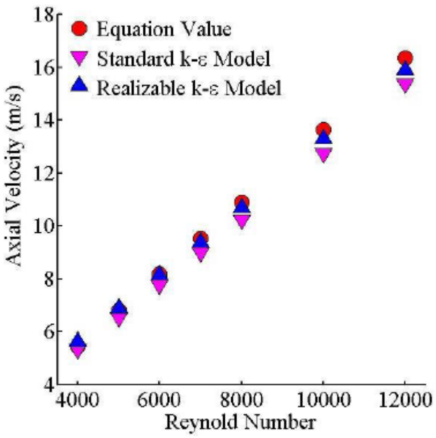

Figure 11: Axial velocity comparison for turbulent flow (at pipe end)

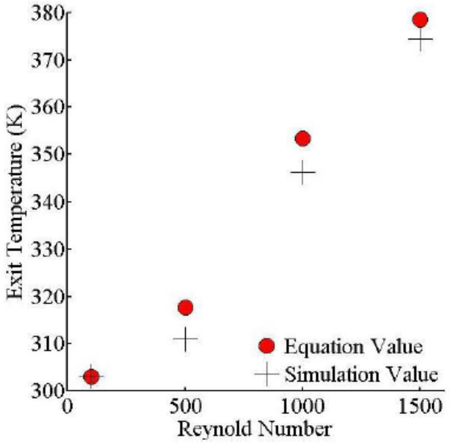

Figure 12: Mean exit temperature comparison for laminar flow (at pipe end)

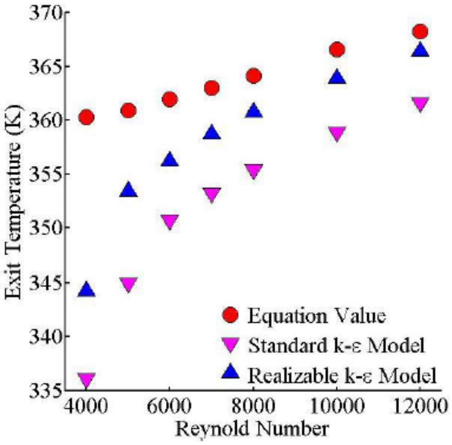

Figure 13: Mean exit temperature comparison for turbulent flow (at pipe end)

4. CONCLUSIONS

Computed axial velocity, mean exit temperature, hydrodynamic entry length, thermal entry length and skin friction coefficient were found in close agreement with the analytical values (3% in laminar and 6 % in turbulent flow). In entry length region, skin friction coefficient, axial velocity, mean exit temperature magnitude changes with length of pipe due to boundary layer development, but remains constant for rest of the length. Axial velocity in developing region increases for certain length of pipe and then it becomes constant, as observed in theory. Opposite to that of axial velocity, the skin friction coefficient against length of pipe reveal that the skin friction decreases along the length of pipe and after some distance it becomes constant, which is observed experimentally. Realizable k-ε turbulent model found more promising in turbulent flow case for both hydrodynamic and thermal calculations.

5. FUTURE WORK

This paper deals with the effect of constant wall temperature on single phase fluid flow in a pipe for different Reynolds number. Further study is needed to find effect of same constant wall temperature on multiphase fluid flow for different Reynolds number.

ACKNOWLEDGEMENT

Authors would like to thank Principle, K. J. Somaiya College of Engineering, Vidyavihar, for providing all the necessary infrastructure and laboratories for carrying out the required work.

REFERENCES

[1] Cengel, Yunus A. Fluid mechanics. Tata McGraw-Hill Education, 2010.

[2] Bhatti, M. S., and R. K. Shah. "Turbulent and transition flow convective heat transfer in ducts." Handbook of single-phase convective heat transfer (1987): 4-1.

[3] Gnielinski, V., Int. Chem. Eng., 16, 359, 1976.

[4] Kays, William Morrow, Michael E. Crawford, and Bernhard Weigand.Convective heat and mass transfer. Tata McGraw-Hill Education, 2012.

[5] Kumar, D. S. Fluid mechanics and fluid power engineering. Katson Publishing House, 1981.

[6] 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

[7] Petukhov, B. S., in T. F. Irvine and J. P. Hartnett, Eds.,Advances in Heat Transfer, Vol. 6, Academic Press,New York, 1970.

[8] Reynolds, Osborne. "An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous, and of the law of resistance in parallel channels." Proceedings of the royal society of London35.224-226 (1883): 84-99.

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

[10] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson Education, 2007.

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

[12] W. Zhi-qing. “Study on Correction Coefficients of Laminar and Turbulent Entrance Region Effects in Round Pipes,” Applied Mathematical Mechanics, 3 (1982), p. 433.