CFD Prediction of Loss Coefficient in Straight Pipes

Abdullah Haroon, Shahbaz Ahmad and Ajmal Hussain

Abstract Loss of head in pipes is an important factor to be considered in design of engineering systems to reduce energy costs. This loss is either due to viscous effect called friction loss or due to change in geometry like bends, elbows, expansion and contraction. The pressure loss is a function of the type of flow of the fluid, i.e., laminar, turbulent; material of the pipe; and the fluid flowing through the pipe. In present study, straight pipes of various materials, namely PVC, steel and cast iron, are analyzed using ANSYS. Equations of mass, momentum and k-ε turbulence model are solved using the finite volume method. To validate the numerical tool friction, loss coefficient is determined and compared with existing experimental results. Furthermore, the effect of variation of Reynolds number on the friction loss coefficient is studied. The contours of turbulence eddy dissipation at various Reynolds number are also presented to investigate the effect of Reynolds number on turbulent kinetic energy.

Keywords Friction loss · CFD · k-ε turbulence model · ANSYS-CFX

1 Introduction

In any pipe assembly, there are two types of resistance against the flow of fluid, namely dynamic losses and frictional losses. Frictional losses are caused mainly due to the shearing stress between the layers of fluid in the laminar sublayer, which are in the vicinity to the surface of the pipe walls. Friction is also caused when the fluid domain in the turbulent flow strikes against the protrusions of the pipe wall. This causes eddy formation and contributes to energy loss. Friction losses take place along the entire length of a pipe. On the other hand, when fluid passes through fitting components such as elbows, curves, tee joints, contractions, expansions,

A. Haroon · S. Ahmad · A. Hussain (✉)
Zakir Husain College of Engineering & Technology, Aligarh Muslim University,
Aligarh, UP, India
e-mail: ajmalamin.iitr@gmail.com

V. Garg et al. (eds.), *Development of Water Resources in India*,
Water Science and Technology Library 75, DOI 10.1007/978-3-319-55125-8_41

exits and entrances, a change in direction or velocity of flow occurs. This change leads to separation of flow from the walls of the pipe and the formation of eddies and turbulence in that area. The loss of energy resulting from these eddies and turbulences constitutes what are called as dynamic losses. Friction losses also known as major losses are unavoidable in any pipe system as they occur throughout the entire length of the pipe and hence very important to keep a track for the estimation of energy losses in the assembly. Since the shear stress of a flow also depends on whether the flow is laminar or turbulent, friction loss also depends on type of flow. For turbulent flow, the losses due to pressure drop depends on the protuberance of the surface, while in laminar flow, the protuberance effects of the walls are insignificant. This is because in turbulent flow, a thin viscous layer (laminar sublayer) is formed close to the pipe wall which causes a loss in energy, while in laminar flow any such layer is absent.

Studies have been performed to get the knowledge about velocity distribution, pressure drop and turbulent flow behavior in the vicinity of rough walls [1–3]. In rough pipes, the inspection of fluctuating velocity spectra is used to find the turbulence profile in all coordinate directions. A significant observation of this study was that the nature of the solid boundary has negligible effect on the flow in the core of the pipe. On the other hand, the flow in the vicinity of the wall is dependent on the nature of the solid boundary [4]. In literature, different approaches were proposed by several researchers to study the relationship of turbulent flow and rough surfaces. The behavior of turbulent flow in pipes by implementing roughness element drag coefficient is studied by Wang et al. [5]. Recently formula for the mean velocity calculation across the inner layer of turbulent boundary is proposed [6]. The velocity profile obtained by busing this formula is used to formulate the friction factor correlation for the fully developed turbulent pipe flow. Saleh [7] observed the effects of roughness by using k-ε turbulence model in conjunction with empirical wall function. Other significant works with the implementation of k-ε turbulence model were studied by Cardwell et al. [8]. K-ε model can be used with moderate roughness within a suitable degree of accuracy. The researchers concluded that among the different turbulence models k-ε model gives the most suitable prediction.

This study focuses on the possibility of employing computational fluid dynamics (CFD) techniques for prediction of friction factor in pipes of various materials. Since every material has a characteristic roughness, friction factor is different for different materials. For the numerical model validation, the Moody's diagram was used [9]. Friction loss coefficient was predicted using standard k-ε turbulence model and was validated from the results appearing in Moody's diagrams. The absolute roughness coefficient for various materials was also read from Moody's diagram. This paper also investigates the effects of Reynolds number on friction loss coefficient and turbulent kinetic energy. ANSYS-CFX is used to solve the RANS equations to achieve convergence residuals of 10E-5.

2 Mathematical Modeling

2.1 Governing Equations

The equation of continuity in differential form can be represented as:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

The momentum equation for the incompressible viscous fluids can be written as:

$$\frac{\partial u}{\partial t}+(u\,.\,\nabla)u=-\frac{1}{\rho}\nabla p+v\nabla^2 u \tag{2}$$

2.2 Turbulence Model

The standard k-ε turbulence model is most famous and simple turbulence model. This model employs the calculation of the turbulent velocity and length scales independently by using the solution of different transport equations. The standard k-ε model has been used widely for solving the turbulence model of practical engineering flow problems [10]. This model is based upon a semiempirical model approach which utilizes model transport equations for the calculation of turbulence kinetic energy k and its dissipation rate ε. The transport equation for turbulence kinetic energy (k) in the model is derived from the exact equation, whereas the transport equation for dissipation rate of kinetic energy (ε) is obtained using physical reasoning and has little resemblance to its mathematically exact counterpart. These equations are as follows:

$$u_i\frac{\partial k}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right)-v_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}-\varepsilon \tag{3}$$

$$u_i\frac{\partial \varepsilon}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right)+c_{1_\varepsilon}\frac{\varepsilon}{k}P-c_{2_\varepsilon}\frac{\varepsilon^2}{k} \tag{4}$$

where

$$P=v_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$$

In the above equations, σ_k, σ_ε, $c_{1\varepsilon}$, $c_{2\varepsilon}$ are empirical constants having standard values as: σ_k = 1.0, σ_ε = 1.3, $c_{1\varepsilon}$ = 1.44, $c_{2\varepsilon}$ = 1.92.

2.3 Other Basic Equations

The Reynolds number is expressed as:

$$Re = \rho UD/\mu, \tag{5}$$

Friction factor (f) can be written as:

$$f = \frac{2\Delta PD}{L\rho U^2} \tag{6}$$

Head loss for fully developed flow is determined by Hagen–Poiseuille's co-relation:

$$H = \frac{fLU^2}{2Dg} \tag{7}$$

3 Numerical Modeling

In the present problem, pipe of length 2 m and diameter 75 mm was used as shown in Fig. 1. The geometry was discretized using a structured hexagonal mesh throughout. O-grid scheme with linearly varying nodal space in the radial direction was used near the wall surface to capture accurately the effect near the walls as shown in Fig. 2. ICEM-CFD was used to generate a 3D geometry and meshing. Material of the pipe was modeled by defining the roughness of walls. Standard roughness of materials was read from Moody's diagram, and the relative roughness was calculated by dividing this value by diameter of pipe as shown in Table 1.

The boundary conditions employed at the boundaries of the domain are as follows:

(1) Boundary condition at inlet: The inlet flow rate is decided by velocity, and the direction is normal to the inlet. The velocities in the other two directions are zero. The temperature is kept as 25 °C.

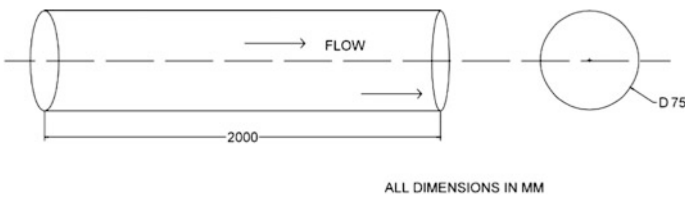

Fig. 1 Schematic view of pipe of length 2 m and diameter 75 mm

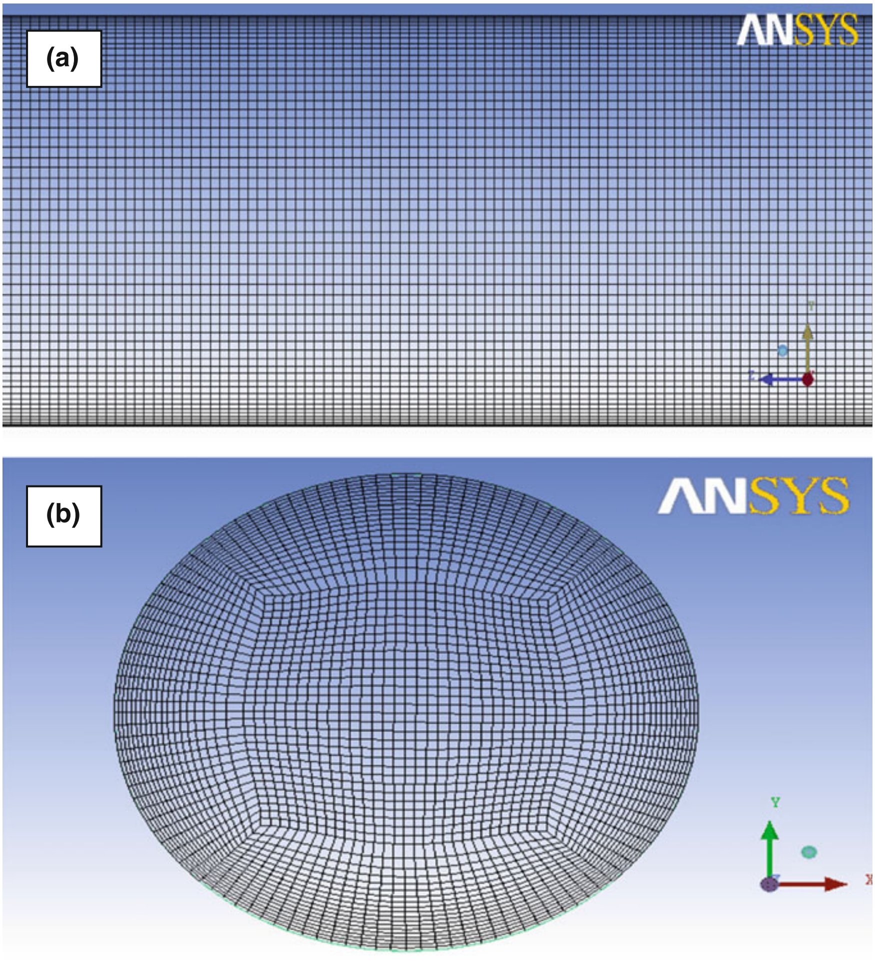

Fig. 2 Meshing of the test pipe **a** side view, **b** inside view

Table 1 Absolute and relative roughness of different materials

Material	Absolute roughness (mm)	Relative roughness (mm)
PVC	0.0025	0.0001
Steel	0.025	0.001
Cast iron	0.15	0.006

(2) Boundary condition at outlet: The outlet boundary condition is kept at zero relative pressure which means that the fluid is flowing out at atmospheric pressure.

(3) Boundary conditions at walls: No-slip wall is chosen as the wall condition. This assumes relative velocity to be equal to zero at the interface of the surface and the fluid, which means that $u = v = w = 0$ at the surface. The wall roughness is altered as per the choice of material.

The governing equations were solved using the finite volume method. The convergence is set such that the scaled residuals decrease to 10E-4 for all the variables. Comparison of friction factor obtained from simulation and Moody's charts is presented in Table 2.

Figure 3 shows the effect of Reynolds number on friction factor of pipes of various materials. It can be inferred from the graph that the friction factor for each pipe decreases as the Reynolds number is increased, which is in corroboration with the friction factor determined from co-relation reported in Moody's diagram.

Table 3 shows the comparison of head loss of this model for PVC pipe with the results obtained by co-relation as shown in Eq. (7). The difference between the co-relation values and simulation results is negligible. Similar comparison can be made for other materials as well.

Table 2 Friction factor comparison for different materials

Material	Absolute roughness (mm)	f (Simulated)	f (Moody's diagram)
PVC	0.0025	0.01788	0.01807
Steel	0.025	0.01914	0.01966
Cast iron	0.15	0.02386	0.02533

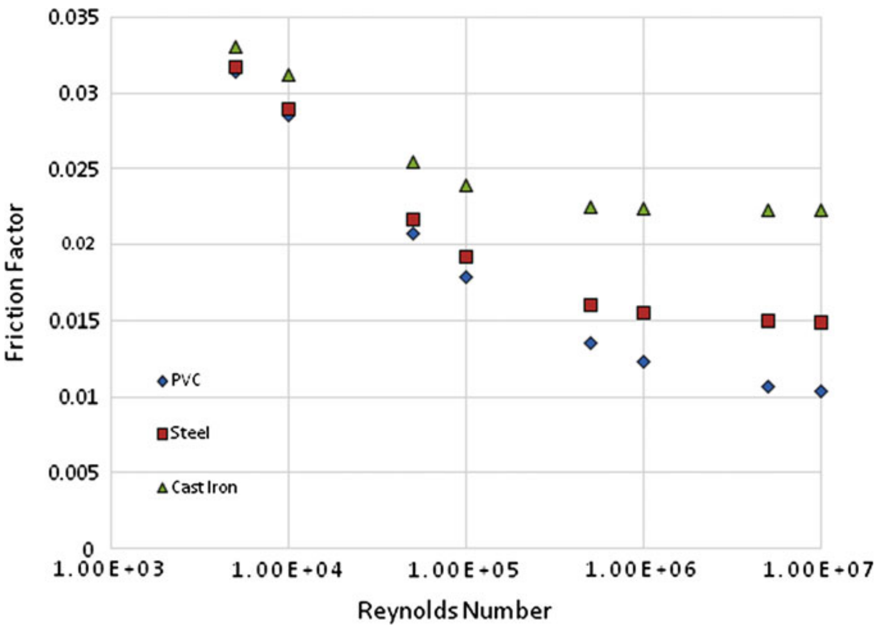

Fig. 3 Effect of changing Reynolds number on friction factor of various materials

Table 3 Comparison of head loss for PVC pipe

Velocity	Head loss (Simulated)	Head loss (Correlation)	% Difference
0.0595	7.54712E-05	7.92015E-05	4.709964895
0.119	0.000274217	0.000279611	1.929346551
0.595	0.004984811	0.005028045	0.85985222
1.19	0.01720683	0.017395463	1.084381008
5.95	0.324415902	0.326237105	0.558245162
11.9	1.186091743	1.1827298	0.284252887
59.5	25.76766565	25.35314611	1.634982641
119	99.94943935	97.77489636	2.224029955

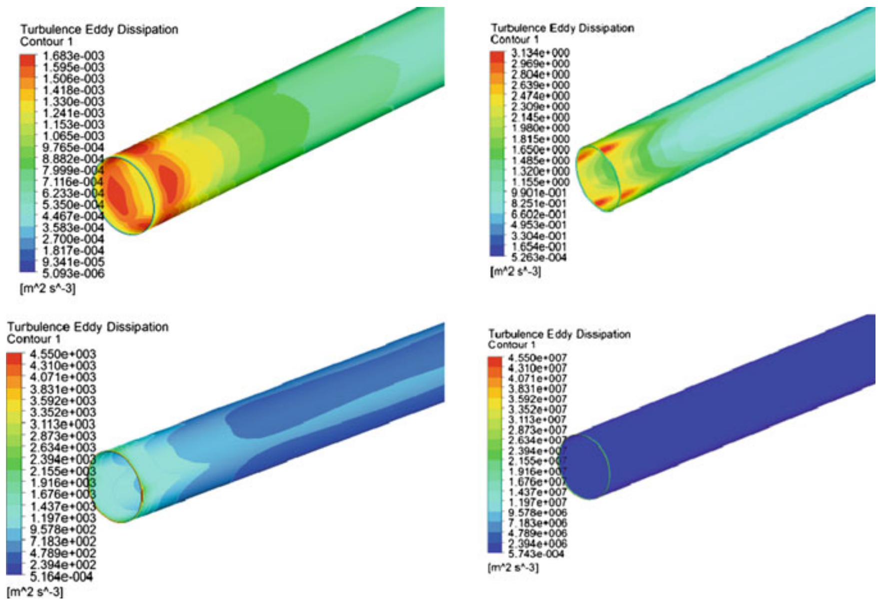

Fig. 4 Effect of different Reynolds number on the turbulence eddy dissipation near the inlet

Effect of different Reynolds number on the turbulence eddy dissipation near the inlet is illustrated in Fig. 4. It is clear from these figures that turbulence at inlet decreases with the increase in Reynolds number.

Figure 5 shows the variation of velocity with Reynolds number at a plane passing through the cross section of the pipe in the middle. In can be observed that as the Reynolds number increases the velocity gradient in the radial direction becomes steeper.

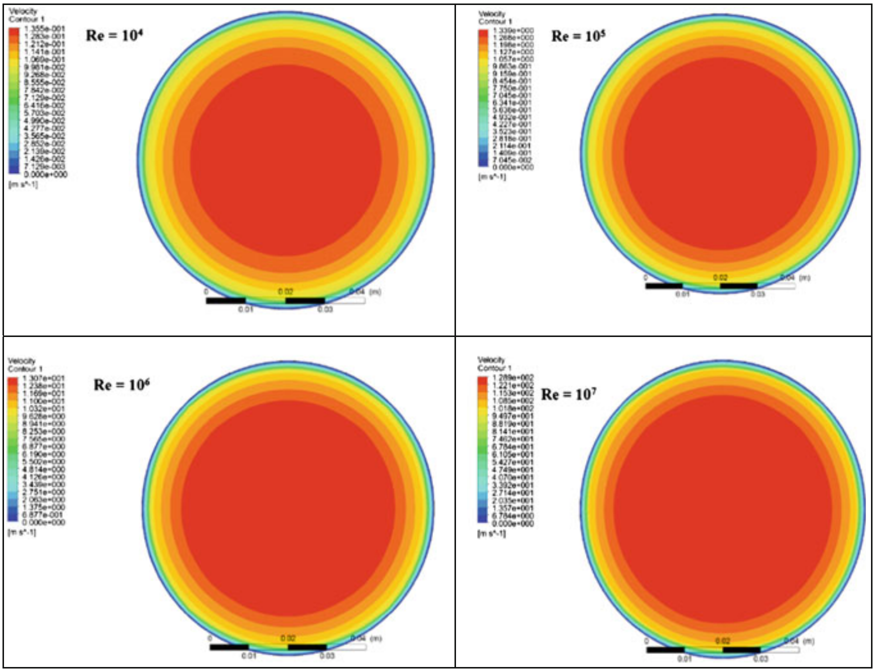

Fig. 5 Variation of velocity with Reynolds number at a plane passing through the cross section of the pipe in the middle

4 Conclusion

In the present study, the extrapolative abilities of the standard *k*-ε turbulence model have been tested as they implemented to the calculation of friction factor of turbulent flow in a 3D pipe with good agreement. This model has been used to predict the various aspects of the fluid flow in a straight pipe, including the wall roughness, friction factor at various Reynolds number and head loss. The advantage of using k-ε turbulence model is that it is computationally cheap. The presented results show a fairly good agreement with the available literature, implying that CFD techniques are mature enough nowadays to predict flow of fluids with standard turbulence models.

References

1. Reif, B.P., Andersson, H.: Prediction of turbulence-generated secondary mean flow in a square duct. Flow Turbul. Combust. **68**, 41 (2002)
2. Vijiapurapu, S., Cui, J.: Simulation of turbulent flow in a ribbed pipe using large eddy simulation. Numer. Heat Transf. Part A: Appl. **51**(12), 1137–1165 (2007)

3. Vijiapurapu, S., Cui, J.: Performance of turbulence models for flows through rough pipes. Appl. Math. Model. **34**(6), 1458–1466 (2010)
4. Rao, A.R., Kumar, B.: Transition of turbulent pipe flow. J. Hydraul. Res. **47**(4), 529–533 (2009)
5. Wang, et al.: Direct simulation of surface roughness effects with RANS and DES approaches on viscous adaptive Cartesian grids. AIAA Paper 2420 (2004)
6. Scibilia, M.-F.: Heat transfer in a forced wall jet on a heated rough surface. J. Therm. Sci. **9**(1), 85–92 (2000)
7. Di Nucci, C., Spena, A.R.: Mean velocity profiles of two-dimensional fully developed turbulent flows. C R Mec **340**(9), 629–640 (2012)
8. Cardwell, N.D., Vlachos, P.P., Thole, K.A.: Developing and fully developed turbulent flow in ribbed channels. Exp. Fluids **50**(5), 1357–1371 (2011)
9. Moody, L.F.: Friction factors for pipe flow. Trans. ASME **66**(8), 671–684 (1944)
10. Rolander, N., Rambo, J., Joshi, Y., Allen, J.K., Mistree, F.: An approach to robust design of turbulent convective systems. ASME J. Mech. Des. **128**(4), 844–855 (2006)