CFD Simulation of Turbulent Flow in Pipes Using Different Turbulent Models

Dr. Alhadi A Abosbaia, Dr. Abdulhafid M. Elfaghi, Dr. Omar A Almatbage
Dept. of mechanical engineering -Faculty of Engineering
Zawia University

Abstract:

In turbulent flow, the local velocities fluctuate in term of magnitude and directions, which will generate small vortices due to viscous shear stresses between neighbouring fluid elements. Those vortices grow in size and merges together resulting in continues mixing of fluid particles hence the momentum will transfer within the fluid. K-omega and K-epsilon turbulence model were used to consider the effects of turbulence of the flow inside a pipe. At the pipe entrance the velocity profile values is equal to the inlet velocity, where the flow is still not fully developed that means velocity distribution still unchanged. This can be clearly seen from the increment of *the centreline velocity values. After the flow is fully developed the centreline velocity reached its maximum value, at the downstream distance of two meter. The velocity profiles along the pipe axis of K-ε model shows more reasonable results with smooth curves than those of the K-ω model. Wall shearing velocity gradients predicted by K-ε model were smaller than that of K-ω model due to the higher Reynolds stress for the earlier model, whereas at the outlet the shearing velocity gradients were smaller for K-ω model.*

Keyword: *Turbulent Flow, K-omega Model, K-Epsilon Model, Centerline Velocity, Velocity Profile.*

1.Introduction:

When the fluid flows at high speed, fluid layers slide on each other, causing a change in the mass of the fluid between the adjacent layers. Such a flow of liquid is known as Laminar Flow [1]. real flow is a mixture of both laminar and turbulent flow, with turbulence flow in center of the pipe whereas the laminar flow near to the edges. Both of these flows behave in different manners in terms of their frictional energy loss though flowing and have completely different calculations that predict their behavior. Transitional flow is a flow that has a Reynolds number in between 2300 and 4000 [2]. As the flow speed increases, fluid layers start moving randomly (transitional flow), and with further increase in flow velocity, the flow of fluid particles becomes completely random and no such laminar layers exist anymore and the flow is turbulent flow when Reynolds number become higher than 4000 [3].

Owing to the presence of the solid boundary the flow behaviour and turbulence structure are considerably different from free turbulent flows. In turbulent thin shear layer flows a Reynolds number based on pipe radius always very large. This implies that the inertia forces are overwhelmingly larger than the viscous forces at these scales [4]. Inertia forces dominate in the flow far away from the wall. Hence along solid boundaries there is usually a substantial region of inertia-dominated flow far away from the wall and a thin layer within which viscous effects are important [5]. Close to the wall the flow is influenced by viscous effects and does not depend on free stream parameters. Far away from the wall we expect the velocity at a point to be influenced by the retarding effect of the wall through the value of the wall shear stress, but not by the viscosity itself [6].

Classical turbulence models (k-e and k-ω models) are based on the presumption that there exists an analogy between the action of viscous stresses and Reynolds stresses on the mean flow. If convection and diffusion are relatively high, hence the dynamics of turbulence must be cosidered. The k-e and k-ω models focuses on the mechanisms that affect the turbulent kinetic energy [7-8].

In present work, K-epsilon and K-omega turbelance models were initially tested for predicting the flow inside of a pipe. The k-epsilon model is very popular for industrial applications due to its good convergence rate and relatively low memory requirements. k-epsilon model does not very accurately compute flow fields that exhibit adverse pressure gradients, strong curvature to the flow, or jet flow, conversely, k-epsilon model does perform well for external flow problems around complex geometries. K-omega model has more difficulty converging and is quite sensitive to the initial guess at the solution. The k-omega model is useful in many cases where the k-epsilon model is not accurate, such as internal flows, flows that exhibit strong curvature, separated flows, and jets.

Numerical Setup:

ANSYSR17.2 coupled with Fluent software were used to predict the pipe turbulent flow properties. The pipe dimension is 0.2 m diameter and 8.0 m length. The inlet velocity is 1.2 m/s, density is 1 kg/m^3 and the viscosity $\mu = 2 \times 10^{-5}$ kg/(ms). Reynolds number based on the pipe diameter and average velocity will be 14700 which is higher than 4000. At this Reynolds number, the flow is usually completely turbulent.

Grid Generation and Boundary conditions:

Structure grid can easily be stretched to account for different flow gradients in different directions. In this cases, the gradients normal to the pipe wall are much greater than tangents ones. Consequently, the cells near the surface have high aspect ratios. The geometry was meshed with 3000 grid points, it is divided into 100 elements in the axial direction and 30 elements in the radial direction. The mesh was clustered in pipe wall vicinity to enhance flow resolution within this important region.

Velocity-inlet, velocity outlet, wall and axis boundary conditions were assigned to the pipe inlet, pipe outlet, pipe wall and the centreline respectively. Grid generation and boundary conditions are shown in figure(1).

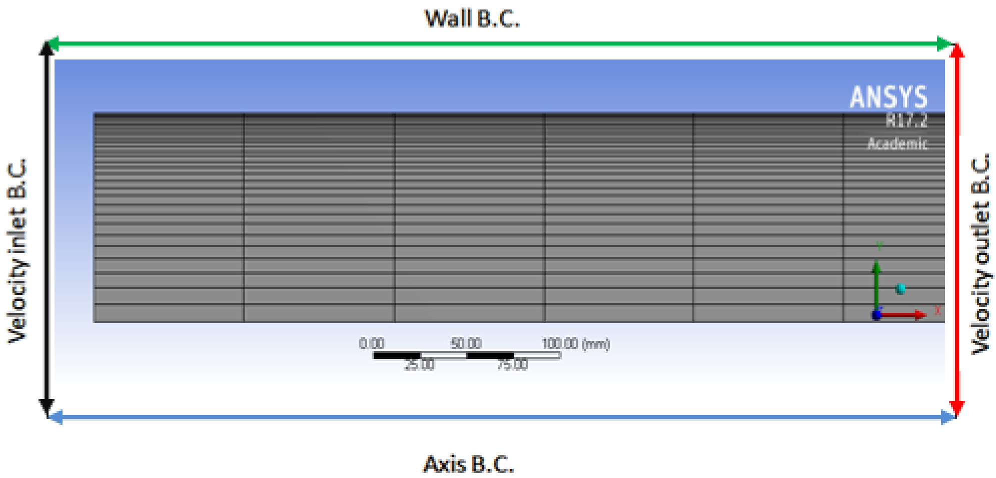

Figure 1: grid generated in the flow domain

Results and Discussion:

Y plus:

Y plus function is examined to determine whether a proper mesh configuration and turbulence model, coupled with near-wall treatment, that lead to accurate computational predictions is obtained.

$$Y^+ = \frac{y}{\mu}\sqrt{\rho\,\tau_w} \qquad \text{……………………(1)}$$

where y is the distance from the wall to the cell centre, μ is the molecular viscosity, ρ is the density of the air, and τ_w is the wall shear stress.

Figures (2), shows wall Y plus distribution along the pipe for K-ε and K-ω turbulence models. Except for few small regions at the pipe inlet Y plus distribution is order of one, that is, the first mesh point away from the wall is placed at a Y-plus of the order of one, which indicates that near wall and the viscous sublayer is acceptable resolved .

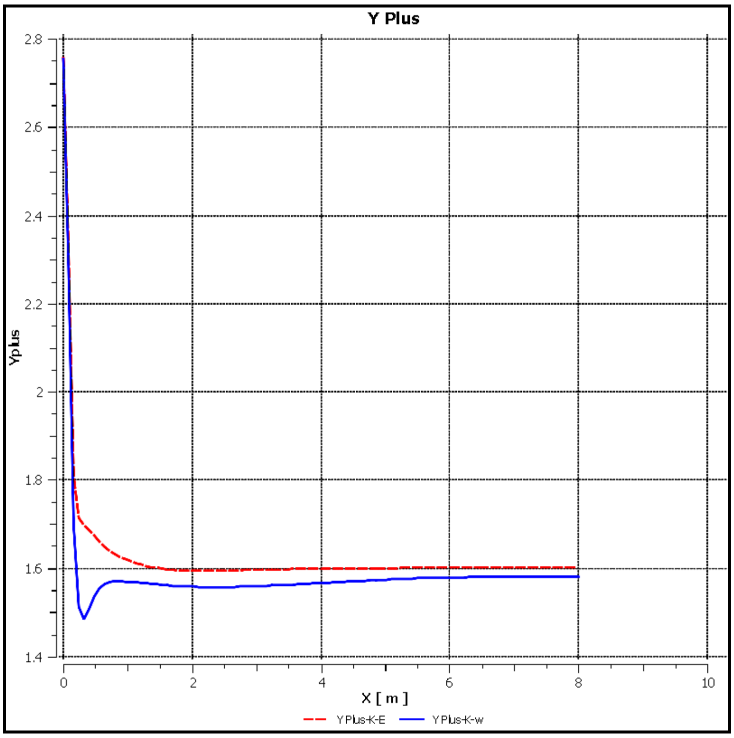

Figure 2: Y^+ function distribution for K-ε and K-ω turbulence models

Velocity Distributions :

For initially undisturbed inlet flow the velocity will be constantly distributed along the pipe in the radial direction except for very thin region near the pipe wall, where the velocity at the wall is equal to zero. In the downstream direction the velocity distribution changes due to the growing boundary layer. Boundary layers continue to grow until the opposite boundary layer meet up at the pipe centreline, hence a fully developed turbulent flow is formed.

At the pipe inlet the centreline velocity is equal to the inlet velocity, the flow is not yet fully developed hence velocity distribution changes. This is clearly seen from the increment of the centreline velocity value. Flow is fully developed and the centreline velocity reached its maximum value at the downstream distance of 10*d, (L=10*0.2m=2m) [9]. Centreline velocity increases beyond the value of 120% of the inlet velocity, this is due the growing boundary layer on both pipe walls in the downstream direction. Velocity distributions along the pipe are shown in Figure (3).

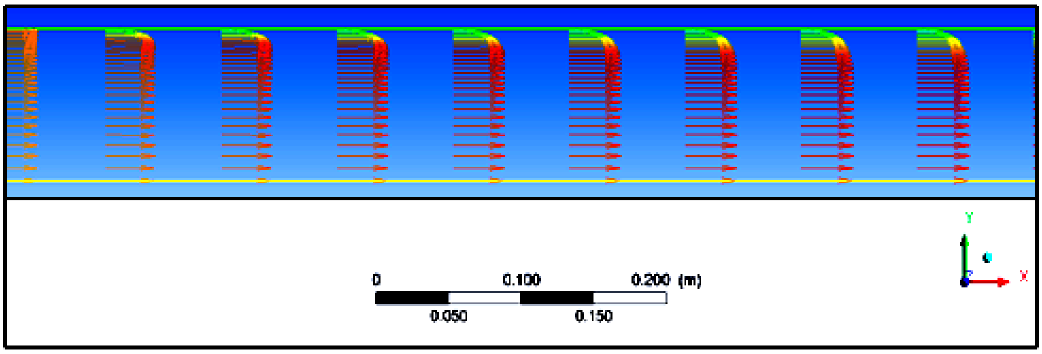

Figure 3: Velocity distribution along the pipe centreline

Centreline Velocity:

Figure (4), illustrates the centreline velocity of pipe flow using K-ε and K-ω turbulent model. For the case of K-ε turbulence model centreline velocity increases within 25% of pipe length to 1.21 times the inlet velocity, subsequently velocity for the rest of pipe is mainly constant at 1.43m/s.

For the case of the K-ω turbulence model, the centreline velocity increases within 43% of pipe length to the peak value of 1.28 times the inlet velocity, further downstream the centre line velocity decreases to 1.24

times the inlet velocity. This indicates that fully developed pipe flow is delayed to 7.5 m of the pipe length.

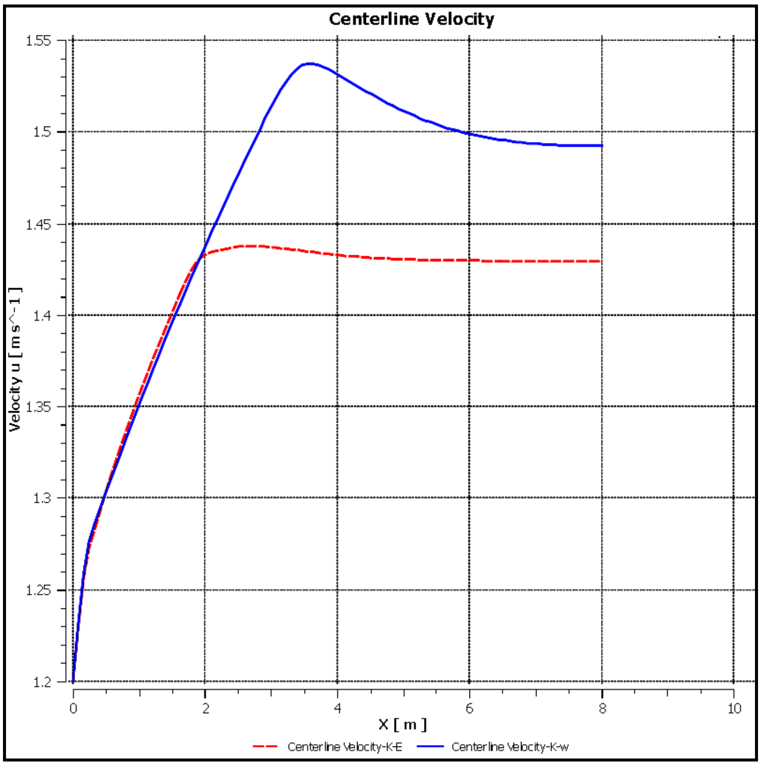

Figure 4: centreline velocity for K-ε and K-ω turbulence models

Coefficient of Skin Friction:

The skin friction coefficient at the pipe wall is predicted using both K-ε and K-ω turbulence models. Skin friction coefficient ($C_f = \frac{\tau_w}{q_\infty}$) starts with the value of 0.0265 at the pipe inlet for both cases. further downstream it decreases to the value of 0.008 within the first half meter from the pipe inlet. Further downstream it remains at this value up to the pipe outlet.

The higher value at the inlet is due to the initial flow disturbances which in turn die quickly and skin friction drops to the value of 0.008. The skin friction distributions is demonstrated in Figure (5).

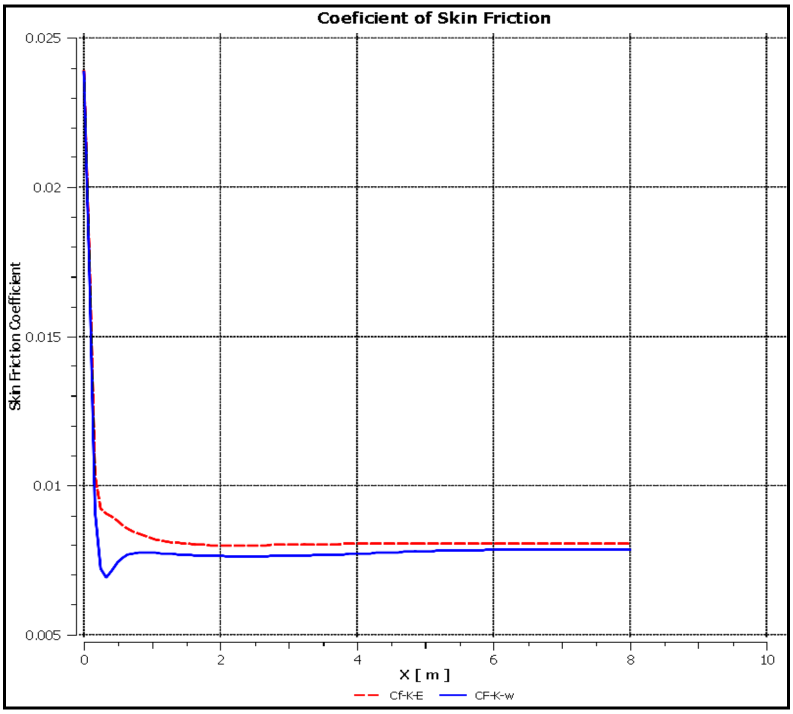

Figure 5: Coefficient of Skin Friction for K-ε and K-ω turbulence models

Velocity Profile:

Figure (6), show the velocity distribution at a given x- station of the pipe for the K-ε and K-ω turbulence models. This is a typical velocity profile of turbulent flow over a smooth wall pipe. The velocity profile has a relatively high slope near to the pipe wall with a quite uniform distribution at and near the centre line. The centreline velocity is higher by 8.4% for the case of k-ω turbulence model. This is reemphasizes the earlier conclusion that the fully developed flow is delayed for the case of K-ω turbulence model.

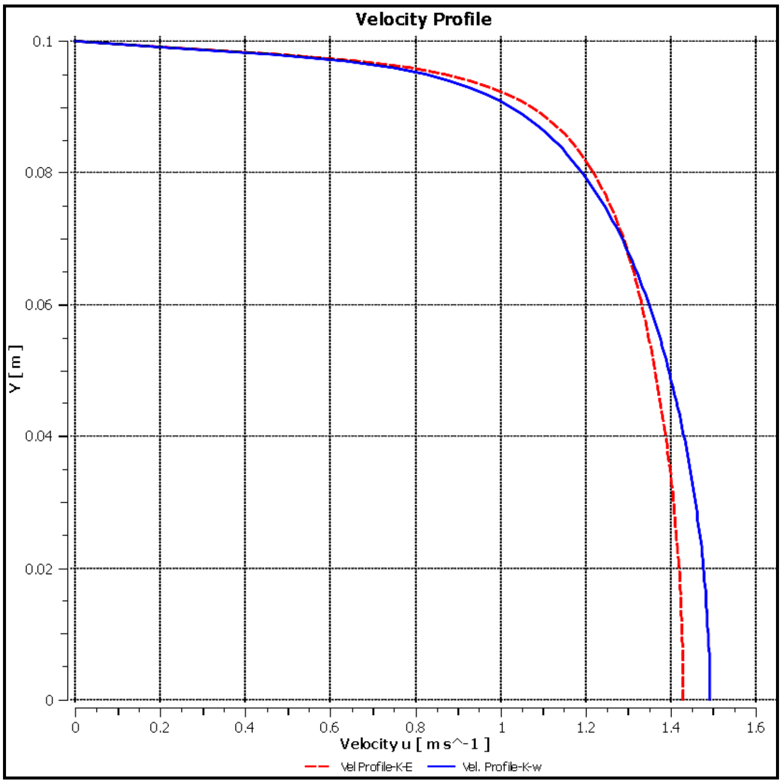

Figure 6 : Velocity profile for K-ε and K-ω turbulence models

Conclusion:

A fluid flowing through a pipe of 8m length at a velocity of 1.2m/s and Reynolds number based on pipe diameter of 14700 is numerically simulated using ANSYS Fluent R17.2, in order to include the effects of turbulence in the analysis, K-ε and K-ω, 2 equation turbulence models were used separately in the solution. Except for few small regions at the pipe inlet Y plus distribution is order of 1 for both turbulence models used. This is an indication of proper near wall resolution. Centreline velocity shows a fully developed flow is obtained earlier in the case of K-ε than the case of K-ω turbulence model. This result was reemphasized in the analysis of velocity profile as well. Results in general will guide to conclude that both turbulence models K-ε and K-ω were capable of properly modeling the turbulence effects in the pipe flow field. Better and more reasonable results were obtained in the case of K-ε turbulence model.

References:

1. *Cengel, Y. A., and Cimbala, J. M. Fluid mechanics: Fundamentals & application. McGraw Hill. 2006*
2. *Andersson, Bengt; et al. (2012). Computational fluid dynamics for engineers. Cambridge: Cambridge University Press. p. 83. ISBN 978-1-107-01895-2*
3. *Anderson D. A, Tannehill JC, Pletcher RH, (1984). Computational Fluid Mechanics and Heat Transfer. McGraw-Hill, New York.*
4. *Anderson, Jr., J. D. (1995). Computational Fluid Dynamics, the Basics with Applications. New York: McGraw-Hill.*
5. *Hoffmann, K. A. & Chiang, S. T. (2000). Computational Fluid Dynamics, Vol. 1 Fourth Edition. Texas, Austin: Engineering Education System.*
6. *ANSYS. 2008. ANSYS Release 15.0 Documentation, ANSYS, Inc., Canonsburg, PA 15317, USA.*
7. *John J. Bertin, Jacques Periaux, Josef Ballmann, Advances in Hypersonics: Modeling hypersonic flows*
8. *Launder, B.E.; Spalding, D.B. (March 1974). "The numerical computation of turbulent flows". Computer Methods in Applied*

Mechanics and Engineering 3 (2): 269–289.doi:10.1016/0045-7825(74)90029-2.

9. *Fluid mechanics, with engineering applications / Robert L. Daugherty, Joseph B. Franzini. 7th ed. McGraw-Hill Kogakusha, 1977.*