A Comparative Analysis of Turbulent Pipe Flow Using $k-\varepsilon$ And $k-\omega$ Models

[1]B. K. Menge, [2] M. Kinyanjui, [3]J. K. Sigey
[1]Department of Mathematics and Physics. Technical University of Mombasa, P.O BOX 90420-80100,Mombasa, Kenya.
[2,3]Department of Pure and Applied Mathematics, Jomo Kenyatta University of Agriculture and Technology, Box 62000-00200, Nairobi, Kenya

***Abstract* - Many if not most flows of engineering significance are turbulent.Fluid engineers need access to viable tools capable of representing the effects of turbulence.In this paper, a computational fluid dynamics model of fully developed turbulent flow in the pipes is implemented with the help of ANSYS FLUENT 6.3.26 software and its preprocessor Gambit 2.3.16. Two Reynolds Averaged Navier Stokes Turbulent models; the $k-\varepsilon$ and $k-\omega$ models are used for the simulation to determine axial velocity, turbulent intensity and skin friction coefficient along the length of the pipe. The Reynolds number based on the pipe diameter and average velocity at the inlet is 10,000.**

The fluid used for this purpose is air and the pipe material is aluminium. The results obtained computationally are compared with experimental data and shows that using the $k-\varepsilon$ model gives more compatible results with those obtained from experiments.

Keywords - Computational fluid dynamics, Reynold's AveragedNavier Stokes,Turbulent intensity, Axial Velocity, Skin friction Coefficient.

NOMENCLATURE

C_f Skin friction coefficient
D Diameter of Pipe, m
I Turbulent intensity
L Length of Pipe, m
$\hat{n}$ outward normal at the surface.
P Pressure
R Radius of pipe, m
Re Reynolds number
S surface of the control volume
U mean velocity
V cell volume $[m^{-3}]$
$\vec{v}$ velocity vector $[ms^{-1}]$
v velocity at the inlet m/s.
τ_w Wall shear stress
ρ Density of fluid kg/m^3
u' root-mean-square of the turbulent velocity fluctuations
μ Dynamic viscosity
∇ gradient operator
τ shear stress [Nm^{-2}]
ε turbulent dissipation $[m^2 s^{-3}]$
k turbulentkinetic energy $[m^2 s^{-2}]$
ω specific dissipation $[s^{-1}]$

1.INTRODUCTION.

All flows encountered in engineering practice become unstable above a certain Reynolds number. At low Reynolds number flows are laminar. At higher Reynolds number, flows are observed to become turbulent. A chaotic and random state of motion develops in which the velocity and pressure change continuously with time within substantial regions of flow. Turbulence is characterized by flow visualization as eddies, which vary in sizes from the largest to the smallest. Largest eddies contain most of the energy, which break up into successfully smaller eddies with energy transfer to yet smaller eddies until an inner scale is reached. In a turbulent flow, the velocity and other flow properties vary in a random and chaotic manner. A turbulent flow can be characterized by the mean values of flow properties and statistical properties of their fluctuations.

In industrial scale, fluid flow patterns are often turbulent and for the prediction of the process, mathematical modelling is needed. Computational fluid dynamics can simulate many processes such as turbulent combustion, heat transfer rate and radiation by using mathematical modelling. Numerical method is the basis of computational fluid dynamics and it is based on mass, energy and momentum continuity equations. First, total fluid space is divided into small components, then the continuity differential equations for each of these components are resolved.

In this paper,the standard $k-\varepsilon$ and $k-\omega$ models were used to model flow in the studied geometry.The Standard $k-\varepsilon$ model (Launder and Spalding, 1974) model is the most widely used complete RANS model and it is incorporated in most commercial CFD codes (Tannehill J. C., Anderson D. A., Richard H. 1997). In this model, the model transport equations are solved for two turbulence quantities i.e. k and ε .The $k-\varepsilon$ turbulence model solves the flow based on the assumption that the rate of production and dissipation of turbulent flows are in near-balance in energy transfer (Ferrey P. and Aupoix B. 2006). The standard $k-\omega$ model(Wilcox,1998) is very similar in structure to the $k-\varepsilon$ model but the variable ε is replaced by the dissipationrate per unit kinetic energy ω.

The main aim of this paper was to obtain a model which gives a better approximation to experimental results obtained from the literature.

2. LITERATURE REVIEW

Several studies have been done on the flow patterns in pipesby Mullin T. and Peixinho J.(2006),Sahu M. *et al*(2009),Willis A. P*et al*(2008).Rudman*et al* (2002) compared results from experimental and numerical investigations of non–Newtonianfluids at transition to turbulence and in weakly turbulent flows. Experimental results showedflow features similar to turbulent puffs and slugs observed in Newtonian transitional flows.Numerical results showed some quantitative discrepancies with the experimental results butdid show turbulence suppression, drag reduction and delayed transition as observedexperimentally.Yogini P. (2010) carried out a numerical simulation of flow past a circular cylinder, using a commercial CFD code (ANSYS Fluent 12.1) with large eddy simulation (LES) and RANSShear-Stress Transport (SST) approaches for Reynolds 1000 and 3900. The numerical results extractedfrom these simulations have good agreement with the experimental data of Zdravkovich M.M. (1997). Analysis of fully developed turbulent flow in a pipe using computational fluid dynamics was carried out by Bhandari D. and Singh S. (2012) and the results obtained computationally were in agreement with analytical results.

3. COMPUTATIONAL DOMAIN

The computational domain and the boundary conditions for the simulation of the flow are shown in Figure 3.1. The cylinder is simulated with a diameter (D) and length L. This paper will consider the flow inside a pipe of diameter 0.25m and a length of 20m using FLUENT 6.3.26. The geometry is symmetric therefore we will model only half portion of the pipe (radius R).

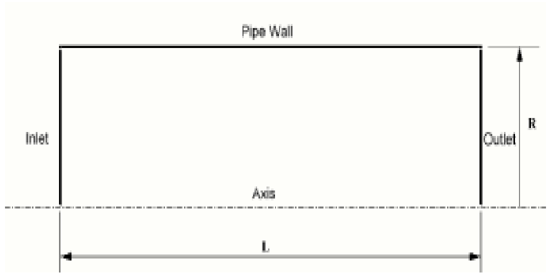


Figure3.1: computational geometry and boundary conditions.

Air enters from the inlet boundary with a constant velocity $u = 1m/s$, density $\rho = 1\ kg/\ m^3$ and coefficient of viscosity $\mu = 2.5\times10^{-5}\ kg/ms$. The fluid exhausts into the ambient atmosphere which is at a pressure of 1 atm. The Reynolds number based on the pipe diameter and average velocity at the inlet is:

$$Re = \frac{\rho u D}{\mu} = 10{,}000$$

At this Reynolds number, the flow is usually completely turbulent.

4. GOVERNING EQUATIONS

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is

$$\frac{\partial\rho}{\partial t} + \nabla\cdot\left(\rho\vec{V}\right) = 0 \qquad (1)$$

and the conservation of momentum equation is

$$\rho\frac{\partial\vec{V}}{\partial t} + \rho\left(\vec{V}\cdot\nabla\right)\vec{V} = -\nabla p + \rho\vec{g} + \nabla\cdot\tau_{ij} \qquad (2)$$

These equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems. However, it is possible to obtain approximate computer-based solutions of the governing equations for a variety of engineering problems. In this investigation,simulation of turbulent flow in a pipe has been done by FLUENT v6.3.26, which uses finite volume approach in which the integral form of the conservation equations are applied to the control volume defined by a cell to get the discrete equations for the cell. The integral form of the continuity equation for steady incompressible flow is

$$\int_S \vec{V}\cdot\hat{n}\,dS = 0 \qquad (3)$$

The integration is over the surface S of the control volume and $\hat{n}$ is the outward normal at the surface. Physically, this equation means that the net volume flow into the control volume is zero.

5. SIMULATION SET-UP

The analysis is carried out with the help of CFD package FLUENT 6.3.26 . Geometry and grid generation is done in GAMBIT 2.3.16 which is the preprocessor bundled with FLUENT.

In this 2D-code, asteady and pressure based solver is used. A least square cell based method is used to calculate gradients. Boundary conditions and the discretization schemes used are summarized in Table 1 below. We have considered convergence criteria 1.0×10^{-6} for these simulations.

Table 5.1: Simulation settings for flow in a pipe with RANS models

Settings	Choice
Simulation type	2D, Steady
Space	Axisymmetric
Solver	Double precision, pressure based, and implicit
Temporal discretization	2nd order
Turbulence model	$k-\varepsilon$ / $k-\omega$ model
Pressure	Standard
Pressure-velocity coupling	SIMPLE
Momentum	2nd order upwind
Turbulent kinetic energy	2nd order upwind
Turbulent dissipation rate (for $k-\varepsilon$ model)	2nd order upwind
Specific dissipation rate (for $k-\omega$ model)	2nd order upwind
Convergence criteria	1×10^{-6}
Boundary conditions:	
Inlet	Velocity inlet
Outlet	Pressure outlet
Top wall	No-slip wall
Bottom wall	Axis

6. RESULTS AND DISCUSSION

Here all simulations have been done using a grid which contains 6000 quadrilateral cells. Characteristics of the simulation in this case are summarized below;

```
Grid Size

Level      Cells     Faces     Nodes    Partitions
    0       6000     12160      6161             1

 1 cell zone, 5 face zones.
```

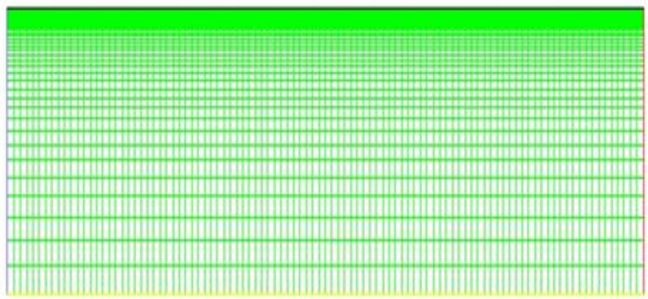


Figure 6.1: Constructed mesh of studied geometry

Numerical results were obtained for the two models for velocity, skin friction coefficient and Turbulent Intensity along the length of the pipe as shown below:

Axial velocity

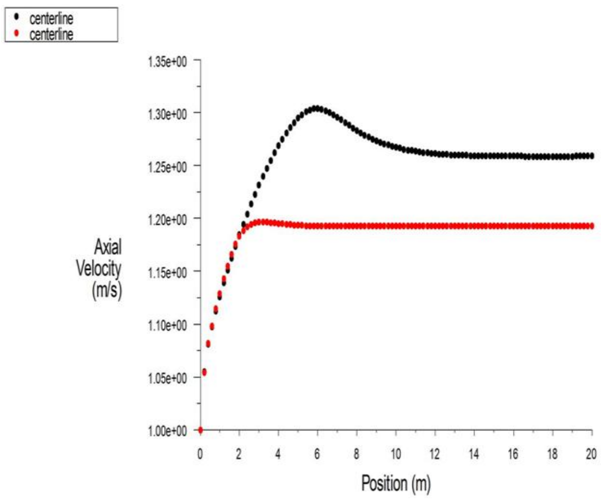


Figure 6.2: Axial velocityalong the centerline of the pipe for $k-\varepsilon$ model (red) and $k-\omega$ model (black)

In figure 6.2, the velocity reaches a constant value beyond a certain distance from the inlet at x=2m

and x = 6m respectively. This is the fully-developed flow region where the centerline velocity

becomes a constant. As the flow develops downstream of the inlet, the viscous boundary layer grows, and will eventually fill the pipe completely (provided that the tube is long enough). When this happens, the flow becomes fully developed and there is no variation of the velocity profile in the axial direction.

Plotting the velocity at the outlet as a function of the distance from the center of the pipe gives

the following results:

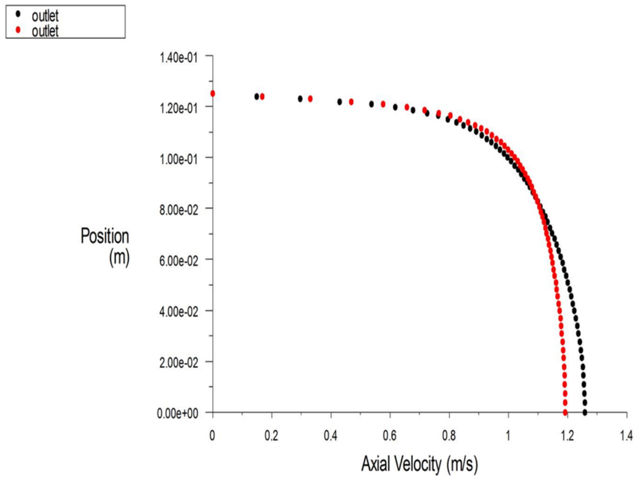


Figure 6.3: Velocity Profileat the outlet fromcenter of the pipe for $k-\varepsilon$ model (red) and $k-\omega$ model (black)

The axial velocity profiles in Figure 6.3 forboth models show a similar trend. They have a flattened shape at the wall corresponding to zero velocity. The vector magnitudes are minimum at the wall and increase towards the centerline .This is in agreement with no-slip condition along the wall and higher velocities at the centerline for conservation of mass. When a fluid is bounded by a solid surface, molecular interactions cause the fluid in contact with the surface to seek momentum and energy equilibrium with that surface. All fluids at the point of contact take on the velocity of that surface. Fluid adjacent to the wall sticks to the wall due to friction effect. The eddy motion loses its intensity close to the wall and diminishes at the wall because of the no-slip condition.

SKIN FRICTION COEFFICIENT

Skin Friction Coefficient is a non-dimensional parameter defined as the ratio of the wall shear stress and the reference dynamic pressure

$$C_f \equiv \frac{\tau_w}{\frac{1}{2}\rho v^2} \tag{6.1}$$

Where τ_w is the wall shear stress, and ρ and v are the fluid density and velocity at the inlet respectively.

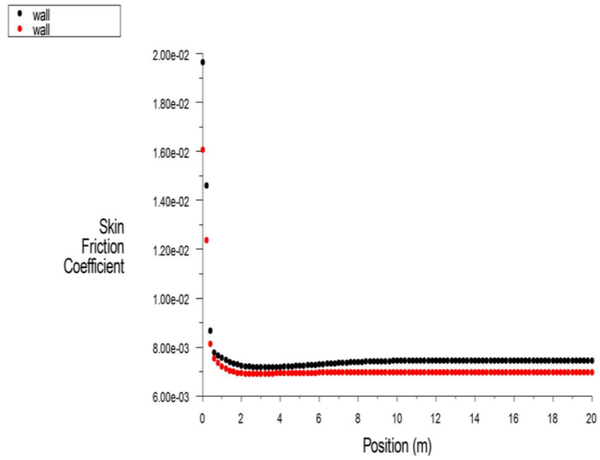


Figure6.4: SkinFriction coefficientalong the top wallfor $k-\varepsilon$ model (red) and $k-\omega$ model (black)

In figure 6.4 for both models, the values at the inlet are much higher and C_f profiles have similar shape all along the length of the pipe. The wall shear stress is the highest at the pipe inlet where the thickness of the boundary layer is smallest and decreases gradually to the fully developed value. Therefore the pressure drop is higher in the entrance region of a pipe resulting in large velocity gradients and consequently larger wall shear stress.

Turbulent Intensity

The turbulence intensity, also often referred to as turbulence level, is defined as:

$$I = \frac{u'}{U} \tag{6.2}$$

Where u' is the root-mean-square of the turbulent velocity fluctuations and U is the mean

Velocity.

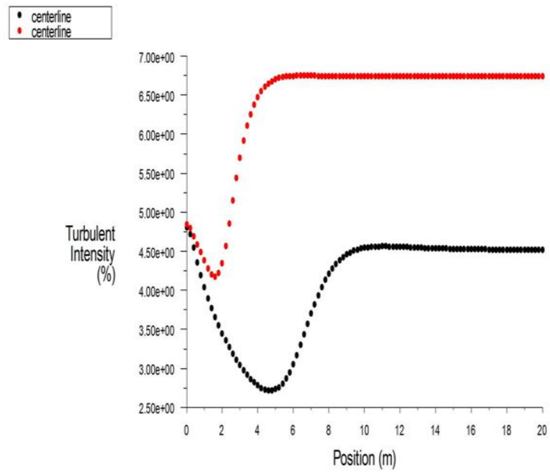


Figure 6.5: Turbulent Intensity along the centerline for $k-\varepsilon$ model (red) and $k-\omega$ model (black)

The turbulent intensities show a decrease at the inlet for both models upto some distance; the first 2m and 6m for the $k-\varepsilon$ and $k-\omega$ modelsrespectively. After these distances there is a rapid increase and the intensity goes beyond the set value of 5% for the $k-\varepsilon$ model. This decrease in turbulent intensity corresponds to an increase in velocity (figure 6.2)

7. CONCLUSION

When compared with experimental results, the centerline velocity for the fully developed region according to figure 6.2 is 1.19m/s with $k-\varepsilon$ model while the experimental results show a value of 1.22m/s. The $k-\omega$ gives a value of 1.26m/s. The results for the first model are closer to the experimental value.

For fully developed turbulent flow of air, the value of skin friction coefficient comes out to be 0.01 while values obtained computationally are 0.0063 and 0.007 for the $k-\varepsilon$ and $k-\omega$ respectively (figure 6.4. The $k-\omega$ has a better prediction for the skin friction coefficient.

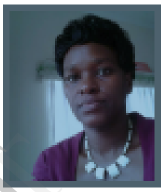
The turbulent intensities show a decrease at the inlet for both models but the prediction of the $k-\varepsilon$ model has significant variation away from the inlet compared to experimental value of 5% (figure 6.5). However it can be seen that a decrease in turbulent intensity corresponds to an increase in velocity.
It is also observed from the results that the axial velocity against position of centerline reveal that the axial velocity increases along the length of the pipe and after some distance it becomes constant which is in conformity with the results obtained experimentally.The axial velocity is maximum at the centerline and zero at the wall to satisfy the no-slip boundary condition for viscous flow

The results of the skin friction coefficient against the wall also reveal that it decreases along the length of the pipe with maximum being at the inlet which is in conformity with experimental results.

REFERENCES

[1] Bhandari D. and Singh S. (2012). Analysis of fully developed turbulent flow in a pipe using computational fluid dynamics. International Journal of Engineering Research & Technology. Vol. 1, 2278-0181, 2012
[2] Ferrey P. and Aupoix B. (2006). Behaviour of turbulence models near a turbulent/non turbulent interface revisited, Journal of Heat and Fluid Flow, Vol.27, p. 831-837.
[3] Launder B. E. and Spalding D. B. (1972). Mathematical models of turbulence, Departmentsof Mechanical Engineering, Imperial College of Science and Technology, London, England.
[4] Mullin T. and Peixinho J.(2006). Transition to Turbulence in pipe flow, Journal of Low Temperature Physics, 145, 75-88.
[5] Rudman M., Graham L. J., Blackburn H. M., Pullum L. *Non-Newtonian Turbulent and Transitional Pipe Flow.*Presented at Hy15, Banff, Canada, 2002
[6] Sahu M., Khatua K. K. ,Kanhu C. P. and Naik T., 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 February 2009, Bhubaneswar, India
[7] Tannehill J. C., Anderson D. A., Richard H., (1997). Computational fluid mechanics and heat transfer, Second addition, Taylor and Francis, London
[8] Wilcox D.C. (1998).Turbulence modeling for CFD, DCW Industries, Inc., La Canada, California.
[9] Willis A. P, Peixinho J., Kerswell R.R., Mullin T. (2008). Experimental and theoretical progress in pipe flow transition, Phil. Trans. Roy.Soc A 366, 2671-2684
[10] Yogini Patel (2010) Numerical Investigation of Flow Past a Circular Cylinder and in a Staggered Tube Bundle Using Various Turbulence Models. *M.Sc* Lappeenranta University of Technology
[11] Zdravkovich, M. M., (1997). Flow around circular cylinders Vol 1: fundamentals, First edition. Printed in Oxford University Press, New York.

BIOGRAPHY



Bathsheba Menge

Bathsheba Menge is a lecturer at Technical University of Mombasa ,department of Mathematics and Physics; has a B.Ed from Kenyatta University (1989) , MSc. Applied Mathematics (2002) from Kenyatta University; and is currently persuing her ph.D in Applied Mathematics Jomo Kenyatta University of Agriculture and Technology.