Analysis of variation of Y+ Value for Fully Developed Turbulent Flow through Pipe using k-ε Turbulence Model

Milap Kumar Sahu1 Brijesh Patel2
1,2M. Tech. Scholar, Assistant Professor
1,2Department of Aeronautical Engineering

School of Engineering & I.T., MATS University, Raipur, (C.G.), India

Abstract— This paper focuses on the analysis of Y+ value for capturing laminar sub-layer along the length for fully developed turbulent flow through pipe. It is important to capture the build out of a fluid flow and head loss in a pipe at higher Reynolds number. A finite element method (FEM) solver with k-ε turbulence model implemented with the help of ANSYS FLUENT 14.5 to scrutinize the flow of water at varying velocities as a result varying higher Reynolds number in a 3D pipe. This paper demonstrates computational investigation of turbulent flow inside pipes for varying inlet velocity. Reynolds Averaged Navier Stokes Turbulent model; the k-ε model is used for the simulation. The Reynolds number varies based upon the inlet velocity. The material of pipe is not the concern. The fluid used for this purpose is water of density 1000 kg/m³. The results acquired are well in agreement with the results acquired experimentally.

Key words: Computational Fluid Dynamics, Reynolds Averaged Navier Stokes, Turbulent, Average Velocity, k-ε Model

I. INTRODUCTION

In recent decades, use of pipes for fluid transmission has great importance globally. Such applications range from the huge, man-made Alaskan pipeline that carries oil almost 1290 km across Alaska, to the more complex natural systems of “pipes” that carry blood through our veins and artery and air into and out of our lungs. The use of pipe transmission system without proper analysis of losses concerned to it has reduced the transmission efficiency. Consequently, demand and attention for efficient flow transmission has increased all over the world. Among new transmission systems, turbulent flow analysis has gained the spot of prime importance.

A pipe is hollow cylinder with dimension of length many times the diameter. Flow through pipes occurs due to pressure gradient, or from higher potential to lower potential. The inner surface of pipe can be smooth or rough, depending on the smoothness of inner surface there is boundary layer formation. The boundary layer formation starts at the solid boundary where variation of velocity occurs from zero to free stream velocity at the center of pipe. Initially the boundary layer formed is laminar boundary layer, as the length of pipe increases this layer increases and becomes unstable which leads to transition from laminar to turbulent flow. The term turbulent flow describes the flow pattern in which fluid flows in haphazard manner, i.e. flow across the layers.

At the entry to a pipe, the fluid develops a boundary layer at the vicinity of pipe walls, while the center of the fluid may remain as an undisturbed uniform flow. Within the boundary layer, viscous stresses are very prominent, hindering the fluid movement due to its friction with the pipe walls. As fluid enters the pipe, layer at the vicinity of pipe walls sticks to pipe surface, reducing velocity at wall to zero. This layer then interacts with next layer and resists its motion, and so on until free stream velocity is reached. Downstream, the boundary layers therefore increases. Eventually, the velocity assumes some average profile across the pipe which is no longer influenced by any means arising from the pipe wall. At this point, the flow is not dependent on what was the effect of pipe wall at the pipe entrance, and we could solve for its properties (such as the velocity profile) without including an entrance region in the calculations.

Fig. 1: fully developed pipe flow [5]

Fig. 2: Turbulent Flow Structure

Turbulent flow in a pipe is a complex episode within fluid mechanics; a lot of researchers have committed determined attempt toward turbulent flow. To capture turbulent flow Y+ value is a necessary parameter to be known. Y+ is a dimensionless distance. It is often used to demarcate coarse and fine mesh for particular flow behavior. It turbulence modeling proper size of the cells near walls is really important. Y+ was introduced by v. Karman for the universal law of the wall for turbulent boundary layers at walls. But it is valid for laminar flows as well. In CFD y+ is used in turbulence models that need the wall distance for modeling the influence of the Reynolds stress tensor.

II. METHODOLOGY

We need to define a proper methodology to carry out the analysis thus shown below the methodology for carrying out the 3D simulation for the analysis of the variation of velocity along the radius for fully developed turbulent flow through pipe.

For the first step various pipe dimension have been selected for various study done. Pipes are classified as smooth pipes or rough pipes based upon the coarseness of the internal surface of the pipe. Smooth pipes are those having very low or no coarseness whereas rough pipes are having high coarseness in internal surface. In this study we have considered smooth pipe.
\begin{align*}
\frac{\partial}{\partial x} (\rho u^2) + \frac{\partial}{\partial y} (\rho u v) + \frac{\partial}{\partial z} (\rho w^2) &= -\frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \\
\frac{\partial}{\partial x} (\rho u v) + \frac{\partial}{\partial y} (\rho v^2) + \frac{\partial}{\partial z} (\rho w v) &= -\frac{\partial P}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial z^2} \right) + g(\rho - \rho_a) \\
\frac{\partial}{\partial x} (\rho w^2) + \frac{\partial}{\partial y} (\rho w v) + \frac{\partial}{\partial z} (\rho u w) &= -\frac{\partial P}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)
\end{align*}

3) Transport Equation for K-ε model

\begin{align*}
\frac{\partial}{\partial x} (\rho k) + \frac{\partial}{\partial y} (\rho \mu \frac{\partial k}{\partial y}) &= \frac{\partial}{\partial x} \left( \frac{\mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x} \frac{\partial k}{\partial y} \frac{\partial k}{\partial z} \right) + G_k + G_b - \rho \varepsilon - Y_m + S_k \\
\frac{\partial}{\partial x} (\rho \varepsilon) + \frac{\partial}{\partial y} (\rho \mu \frac{\partial \varepsilon}{\partial y}) &= \frac{\partial}{\partial x} \left( \frac{\mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x} \frac{\partial k}{\partial y} \frac{\partial k}{\partial z} \right) + C_{1 \varepsilon} \frac{\mu}{k} G_k + C_{2 \varepsilon} \rho \varepsilon^2 + S_\varepsilon
\end{align*}

\begin{align*}
\mu &= \rho C_\mu \varepsilon \\
C_\mu &= 0.09 \\
\sigma_k &= 1.00 \\
\sigma_\varepsilon &= 1.30 \\
C_{1 \varepsilon} &= 1.44 \\
C_{2 \varepsilon} &= 1.92
\end{align*}

Mesh generation: For the analysis a hex mesh has been generated. Sweep method was followed in order to sweep the mesh elements from start to end. Velocity inlet was chosen. The dimensions of the hex mesh are as follows.

<table>
<thead>
<tr>
<th>Edge sizing</th>
<th>32 parts</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inflation</td>
<td>10 layers</td>
</tr>
<tr>
<td>Thickness of layer</td>
<td>5 mm</td>
</tr>
<tr>
<td>Elements</td>
<td>210120</td>
</tr>
<tr>
<td>Nodes</td>
<td>862458</td>
</tr>
</tbody>
</table>

Table 2: Hex Mesh Conditions

A. Mathematical Modeling and Assumption

1) Assumptions:
   - Steady state and Turbulent flow (Reynolds number is >2000)
   - Negligible heat transfer (Since we are concentrating on CFD flow analysis through pipe)
   - Incompressible fluid
   - Fully Developed turbulent flow

B. Various equations used are as follows

1) Continuity Equation

\[ \frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) + \frac{\partial}{\partial z} (\rho w) = 0 \]  

2) Momentum Equation

Let (x, y, z) be the orthogonal components of the body force field in the Cartesian coordinate system; then.

C. Convergence

Due to the iteration performed in simulation we use convergence as the monitor for achieving the final solution. The criterion of convergence of this numerical solution is
Analysis of variation of Y+ Value for Fully Developed Turbulent Flow through Pipe using k-ε Turbulence Model

Based on the absolute normalized residuals of the equations that were summed for all cells in the computational domain. Convergence was considered as being achieved when these residuals became less than $10^{-5}$, which was the case for most of the dependent variables. Iterative convergence was also checked when values of velocity, continuity, kinetic energy and turbulent dissipation became almost constant. Furthermore, checks for the achievement of a final solution were made on the basis of the conservation of mass.

**III. RESULTS**

The friction factor in the pipe has been calculated using FVM solver ANSYS Fluent 14.5 for the velocities 0.01 m/s, 0.015 m/s, 0.02 m/s, 0.025 m/s, 0.03 m/s.

<table>
<thead>
<tr>
<th>Inlet velocity m/s</th>
<th>Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.010</td>
<td>10000</td>
</tr>
<tr>
<td>0.015</td>
<td>15000</td>
</tr>
<tr>
<td>0.020</td>
<td>20000</td>
</tr>
<tr>
<td>0.025</td>
<td>25000</td>
</tr>
<tr>
<td>0.030</td>
<td>30000</td>
</tr>
</tbody>
</table>

Table 3: Inlet Conditions

**Fig. 6: Convergence of Solution**

After the analysis is done with selected model we get the variation of Y+ value along length for fully developed turbulent flow and the parameter friction factor in the pipe which has been used to validate the results from the base work.

**Fig. 7: Y+ value contour for Reynolds number 10000**

**Fig. 8: Y+ variation for Reynolds number 10000**

**Fig. 9: Y+ value contour for Reynolds number 15000**

**Fig. 10: Y+ variation for Reynolds number 15000**

**Fig. 11: Y+ value contour for Reynolds number 20000**

**Fig. 12: Y+ variation for Reynolds number 20000**

**Fig. 13: Y+ value contour for Reynolds number 25000**

**Fig. 14: Y+ variation for Reynolds number 25000**
Analysis of variation of $Y^+$ Value for Fully Developed Turbulent Flow through Pipe using $k-\varepsilon$ Turbulence Model

(IJSRD/Vol. 4/Issue 05/2016/207)

Fig. 14: $Y^+$ variation for Reynolds number 2500

Fig. 15: $Y^+$ value contour for Reynolds number 30000

Fig. 16: $Y^+$ variation for Reynolds number 30000

After performing the iteration for various Reynolds number value of friction factor, shear stress and head loss is obtained.

A. Friction Factor

In fluid dynamics, the Darcy friction factor formulae are equations that allow the calculation of the Darcy friction factor, a dimensionless quantity used in the Darcy–Weisbach equation, for the description of friction losses in pipe flow as well as open-channel flow.

<table>
<thead>
<tr>
<th>Friction factor (simulated)(1)</th>
<th>Friction Factor (literature)(2)</th>
<th>Difference (1)-(2)</th>
<th>Reynolds Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0320</td>
<td>0.0310</td>
<td>0.0010</td>
<td>10000</td>
</tr>
<tr>
<td>0.0284</td>
<td>0.0280</td>
<td>0.0004</td>
<td>15000</td>
</tr>
<tr>
<td>0.0280</td>
<td>0.0270</td>
<td>0.0010</td>
<td>20000</td>
</tr>
<tr>
<td>0.0256</td>
<td>0.0248</td>
<td>0.0008</td>
<td>25000</td>
</tr>
<tr>
<td>0.0248</td>
<td>0.0237</td>
<td>0.0011</td>
<td>30000</td>
</tr>
</tbody>
</table>

Table 4: Comparison of friction factor obtained by simulation and reference work

Fig. 17: Friction factor versus Reynolds number

The result shows that the simulation is well validated with the reference work.

B. Shear Stress

Wall shear stress remains constant throughout the pipe during the flow. For various frictions factor shear stress varies as.

Fig. 18: Friction Factor versus Average shear stress

It is evident from above plot as friction factor is varied with average shear stress it can be observed that friction factor decreases with increase in shear stress since Reynolds number is increasing so along the wall shear stress increases decreasing the friction factor.

C. Head Loss

The head loss in turbulent flow in a pipe varies as (velocity)

Fig. 19: Head Loss versus Velocity

As the velocity increases along the pipe length Reynolds number increases, head loss begins to increase due to increase in average shear stress along the pipe wall.

IV. DISCUSSION

In this paper we've obtained variation of $Y^+$ value, friction factor, average shear stress, head loss. It is clear that $Y^+$ value is high at the entry of pipe and becomes constant after few distances from the entry. Friction factor decreases with
increase in inlet velocity, which means at high speed effect of friction is less. Head loss for varying inlet velocity increases with the increase in velocity. When friction factor and average shear stress is compared we came to know friction factor decreases with increase in average shear stress.

REFERENCES


