Journal of Multidisciplinary Engineering Science and Technology (JMEST)
ISSN: 2458-9403
Vol. 7 Issue 3, March - 2020

Analysis of Water Flow in Tee-junction Pipes Using CFD

Okafor C.V[1]
[1] Department of Building,
Nnamdi Azikiwe University Awka,Nigeria.
Chinedumokafor117@yahoo.com

Ononye O.R[2]
[2]Department of Building Technology Education
Federal College of
Education(Technical),Asaba,Delta State of Nigeria.
royrovas@yahoo.com

Okeke A.U[3]
[3]Department of Building,
Nnamdi Azikiwe University Awka,Nigeria.
brownvilley@yahoo.com

***Abstract*— the purpose of this paper is to analyze steady state incompressible flow of water through a T-junction pipe using computational fluid dynamics simulation. Turbulent flow regime with Reynolds number equal to 224045 was investigated in study. The geometry of the Tee-junction pipe for the study has a diameter (D) of 0.2m and Tee length of 5D. From the numerical analysis, it was observed that velocity of water flow is strongly influenced by 90 degrees angle of T-joint of pipe resulting to minor losses.**

Keywords—Tee junction pipe; Minor Loss; CFD;RNG K-ε Model.

I. INTRODUCTION

Fluids are transported through flow sections. Flow sections of circular cross sections are known as pipes. Most fluid especially liquids are transported in circular pipes. This is because, pipes with a circular cross section can withstand large pressure differences between the inside and outside without significant distortion. A typical piping system involves pipes of different diameters connected to each other by various fittings (T-Junction, Y-Junction) or elbow to ensure continuous transport of the fluid.

Fig 1: Tee-Joint

Frictional losses occur in pipe flow. This may be due to friction between the fluid particles (viscosity) or between the fluid particle and the wall shear stress (No slip condition). The frictions are related to pressure drop and head loss during flow through pipes. According to [1], energy input is needed to make the fluid flow through the pipe because of frictional energy loss (frictional head loss and frictional pressure drop) due to the friction between the fluid and the pipe wall.

Minor loss is a term used to describe losses that occur in fittings, expansions, contractions and the like. Fittings commonly used in the industry include bends, Tees, elbows, unions and valves used to control the flow [2].

In this work, the researcher focuses on analyzing the minor losses occurring in a common component of pipe network “T-Junction”.

II. Numerical Setup

The geometry model, which is studied consist of a T-Junction pipe with a diameter of 0.2m. The length of the pipe is equal to 5 times the Diameter (L=5D). The discritiation of the 3D model was performed in LISA by using quadratic shell element supplied by nine nodes, six degree of freedom per nodes and quadratic interpolation function with reduced integration. The element has three of shape functions which are associated with corner nodes, middle nodes and centre nodes respectively.

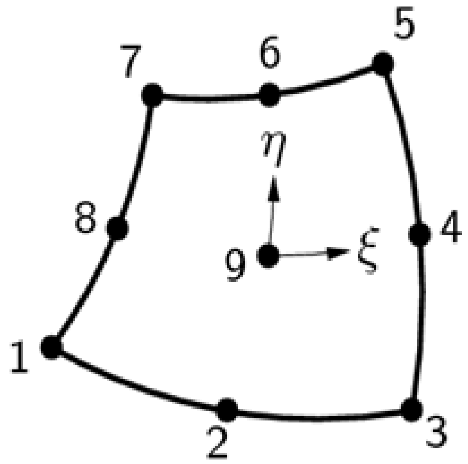

Figure 2: Nine Node Biquadratic Element

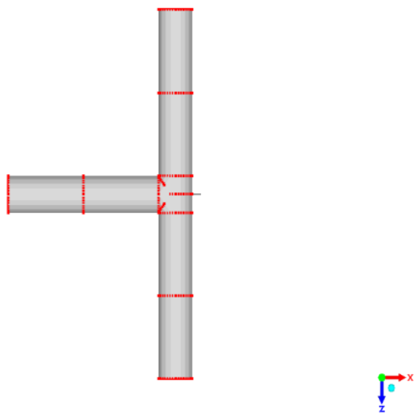

Figure 3: FEA Model of the Te-junction Pipe showing Nodes and element surface

Two cases are considered, in the first case, the outlets is considered at an angle 90 degrees to the inlet while in the second case, the outlets are at 180 and 90 degrees respectively to the inlet.

III. Boundary Condition

The boundary condition shall be determined in order to provide information about the velocity fields and its gradient at the natural boundary of the flow domain. These boundary conditions are used in solving the governing equation for the velocity and pressure field. Fig 4 and 5 shows the schematics of the boundary condition of the Tee-pipe for both cases.

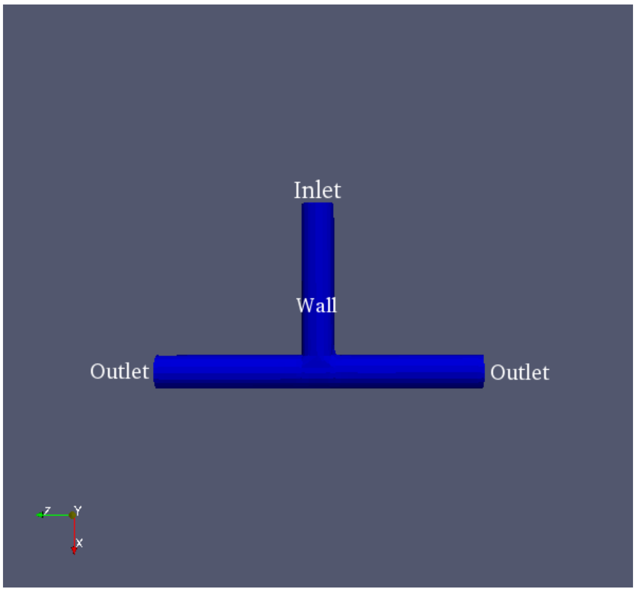

Figure 4: Schematic of Boundary Condition of the Pipe for Case 1

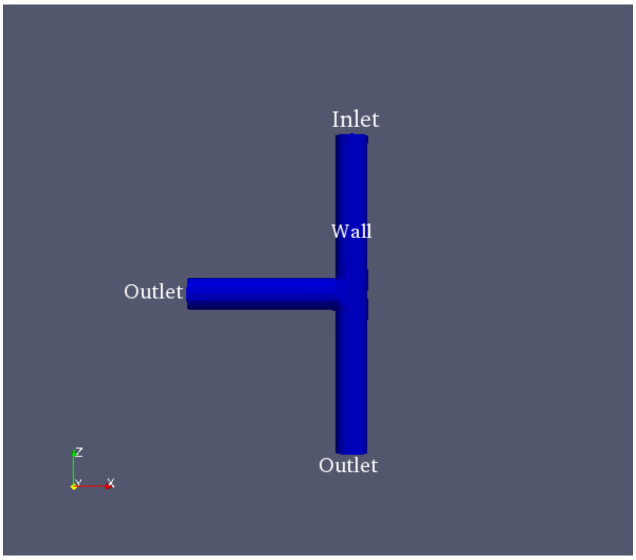

Figure 5: Schematic of Boundary Condition of the pipe for Case 2

A. *Inlet Boundary Condition*

The inlet distribution of k and E in the internal flow was obtained from the turbulent intensity T, and a characteristic length (L) of the equivalent pipe radius by means of the following assumed form

$$K = 3/2(U_{ref}.T)^2 \quad (1)$$

$$E = C_\mu^{3/4} . \frac{K^{3/2}}{L} \quad (2)$$

Where L= 0.07l

B. *Wall Boundary Condition*

At the wall, a no-slip condition was adopted at the wall. Velocity was set at zero. Pressures are set at zero gradients at every direction.

$$U_x = 0,\ U_y = 0,\ U_z = 0 \quad (3)$$

$$\frac{\partial p}{\partial x} = 0, \frac{\partial p}{\partial y} = 0, \frac{\partial p}{\partial z} = 0 \quad (4)$$

C. *Outlet Boundary Condition*

At the outlet, the velocity is set to be zero gradient for every direction and pressure is set to zero.

$$\frac{\partial u}{\partial x} = 0, \frac{\partial v}{\partial y} = 0, \frac{\partial w}{\partial z} = 0. \quad (5)$$

$$P_x = 0; P_y = 0; P_z = 0. \quad (6)$$

IV. SOLVER SETTING

Simple algorithm was used for pressure-velocity coupling, pressure interpolation was second order and second order discritization scheme was used for both the convective term and viscous term of the governing equation for fluid flow.

Simple algorithm which stands for semi-implicit method for pressure linked equation generates the pressure from velocity components by utilizing the Navier Stokes Equation combined with iterative procedure [3]. Steady state analysis used to develop the adaptive mesh carried out using an RNG K-ε turbulence model by [4].

V. RESULT AND DISCUSSION

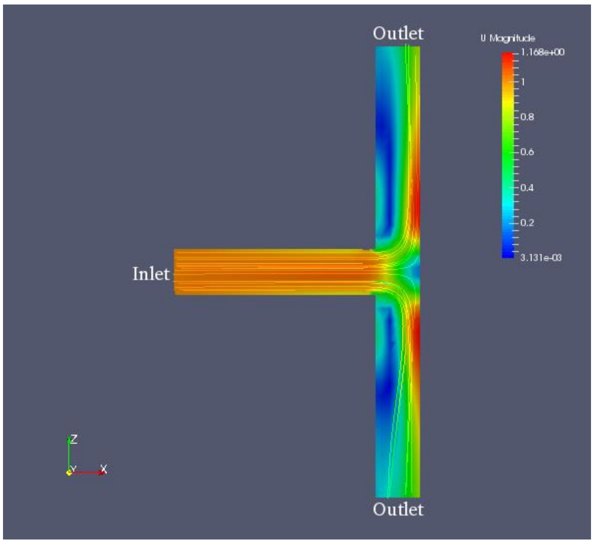

Fig 6: Velocity Contour in m/s for case 1

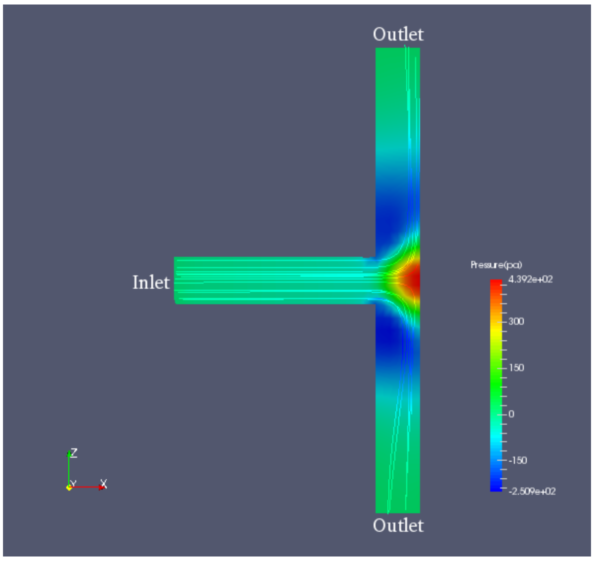

Fig 7: pressure contour for case 1

By plotting the streamline of the fluid flow inside the pipe, the behavior of the fluid in the pipe could be understood. The different color of the streamline represents different contour of velocity magnitude.

Fig 6 and 7 above shows the pressure and velocity contour of pipe for case 1. The color contour shows that pressure increases at the connecting branch of the pipe at a value greater than 400pa due to which pressure loss occurred. The pressure loss in the pipe was also accompanied by a decrease in fluid velocity as seen in fig 7.

As shown in fig 7, the velocity increases at the inlet of the pipe up to an approximate value of 2m/s. This fluid velocity immediately tends to zero as it approaches the connecting branch (No Slip) creating pockets of turbulence eddies at the corners resulting in dissipation of mechanical energy into intermolecular energy thus energy loss is experienced.

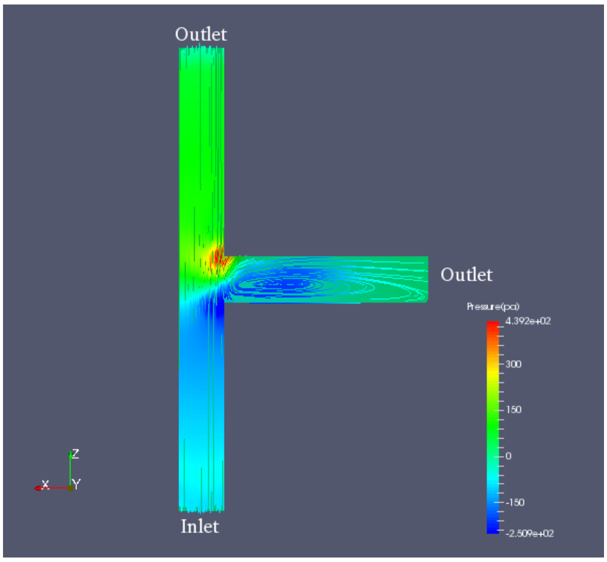

Fig 8: Velocity contour in m/s for case 2

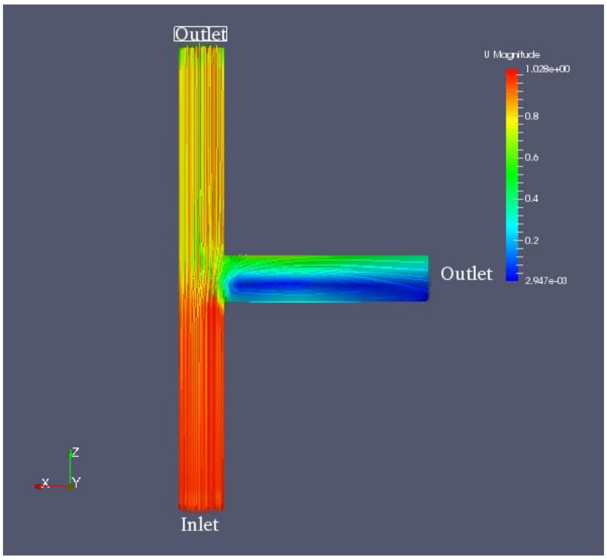

Fig 9: Pressure contour for case 2

From the velocity contour diagram shown in fig 8, it can be deduced that fluid velocity increased at the inlet as it approaches the branched connection which in turn results to a reduction in flow velocity accompanied with pressure loss at that region (the connecting branch). As the fluid tends to negotiate with the connecting branch, a series of recirculation flow occur causing a reduction in the velocity of the flow as it reaches the outlet.

VI. CONCLUSION

In this paper, three- dimensional numerical simulation of the fluid flow in the T-Junction pipe was presented. From the result of the analysis of the physical flow obtained, the main conclusion of the present work shows that the velocity of water flow is strongly influenced by the T-Joint of the pipe.

Both case 1 and case 2 experienced pressure losses which were as a result of the no-slip condition encountered when the fluid tends to negotiate the Tee-junction. Recirculation flows were also observed in case 2 which further increases the pressure loss at that region.

REFERENCES

[1] G.E Nimadage, and S.V Chopade, “CFD analysis of flow through T-junction of pipe,”. International Research Journal of Engineering and Technology.Vol 4,No 2,pp. 906-911, 2017

[2] R.S Shankar, Pipe flow Calculations. Department of chemical and bio molecular engineering, Clarkson University

[3] J. Ferziger, and M. Peric, Computational Methods for fluid Dynamics. 2nd ed.s.l; Springer,1999

[4] V.Yakhot,S.Orszag,S.Thangam,T.Gatski,C.Speziale,“Development of turbulence model for shear flows by a double expansion technique” physics of fluids.Vol.4,No.7,pp. 1510-1520,1992.