International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 04 Issue: 06 | June -2017 www.irjet.net p-ISSN: 2395-0072

Investigation of flow parameters and structural analysis of Y-Duct

Sunil Kumar R[1], Prof. U. B. Khadabadi[2]

[1]Department of Mechanical Engineering (M.Tech in Design Engineering)
KLE Dr. M S Sheshgiri College of Engineering and Technology Udyambag, Belagavi, Karnataka, India 590008
[2]Department of Mechanical Engineering (Professor)
KLE Dr. M S Sheshgiri College of Engineering and Technology Udyambag, Belagavi, Karnataka, India 590008

Abstract - *Pipe networks consists of Y-junction, elbows, T-junctions, bends, contractions, expansions, valves and many other components. These components cause loss in pressure due to change in momentum of the flow caused due to friction and pipe components. In fluid flow analysis, the geometry with branch angle plays an important role in the distribution of flowing fluid, pressure and velocity of fluid. The purpose of maintaining branch angle is to distribute the flowing fluid with proportion volume through branched segments. But larger angle variation may be the reason to build uneven distribution, loss of pressure, velocity variation and turbulence formation.*

In general, the effect of branching in pipe network is the complication in flow patterns. Change in angle will also affect the other flow parameters like, flow distribution, pressure etc. In this investigation an attempt is made to evaluate the flow parameters which affect the distribution of flow at the junction, pressure loss by comparing the various branching angle results. The small junction of pipe network i.e. Y branch is been concentrated in the present investigation. The mesh tool (Altair Hypermesh 16.0) had been used to model the geometry and to mesh the same with different branching angle. Further, the simulation part is carried using Ansys 16.0 (Fluent). The results are evaluated for different velocities by changing the branch angles. Though this investigation it had been concluded that, pressure drop with respect to velocity variation is high at bent angle of 45^0 compare to 60^0, 90º, and 180 º. Also, the pressure distribution at the two outlets is uniform for an angle 45^0 compare to 60^0, 90^0 and 180^0. This uniform pressure distribution at the outlets is due to the least effect of the turbulence for 45º branching. The flow distribution of fluid at the junction is uniform for an angle of 45^0 compare to 60^0,90^0 and 180^0.

This uniformity in distribution of flowing fluid is the result of less angle segment and turbulence at the junction. It can be conclude that as bent angle goes on increases, it will directly effect on pressure drop, pressure distribution and distribution of flowing fluid. The structural analysis is carried out for a bent angle 45º with preferred velocity. The same pressure developed inside the pipe due to fluid flow is made to act on the inner surface of the pipe. From this analysis the effect of pressure developed on the pipe-wall had been studied and the preferred condition is suggested.

1. INTRODUCTION

The flow of liquids through pipe networks is used in industries as well as the households. The supply of water for household purposes involves the transport of water through pipes which have small diameters. In industries the water and other fluids supply is done through pipelines with comparatively larger diameter. The mechanics of the flow of fluids depends on the properties of the fluids being transported. These fluid properties include density, viscosity and surface tension.

Pipe networks are mainly used for transportation and supply of fluids and gases. These networks vary from fewer pipes to thousands of pipes (e.g. water supply network of a large city). In addition to pipes, the network also consists of Y-junction, elbows, T-junctions, bends, contractions, expansions, valves, meters, pumps, turbines and many other components. All these components cause loss in pressure due to change in momentum of the flow caused due to friction and pipe components. This means conversion of flow energy in to heat due to friction or energy lost due to turbulence. Pipe networks are very common in industries, where fluid or gases are to be transported from one location to the other. The head loss (pressure loss) may vary depending on the type of components occurring in the network, material of the pipe and type of fluid transported through the network. In industries the networks are usually large and require very precise pressure at certain points of network. It is also sometimes essential to place valves, pumps or turbines of certain capacity to control pressure in the network. The placement of valves, pumps and turbines is important to overcome pressure losses caused by other components in the network.

1.1 Branching Angle and Its Effects

In fluid flow analysis, the geometry with branch angle plays an important role in the distribution of flowing fluid, pressure and velocity of fluid. The purpose of maintaining branch angle is to distribute the flowing fluid with proportion volume through branched segments. But larger angle variation may be the reason to build loss of pressure, velocity variation and turbulence formation. In general, branching flow effect the flow patterns and makes it complicated, hence flow modeling and analysis at the branching regions.

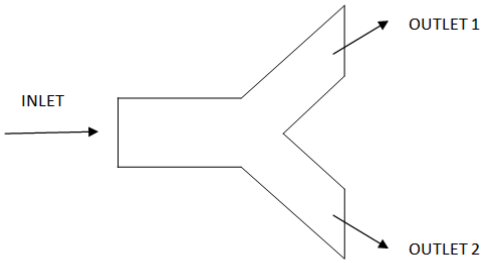


Fig-1: Schematic Diagram of Y-Duct

2. METHODOLOGY FLOW CHART

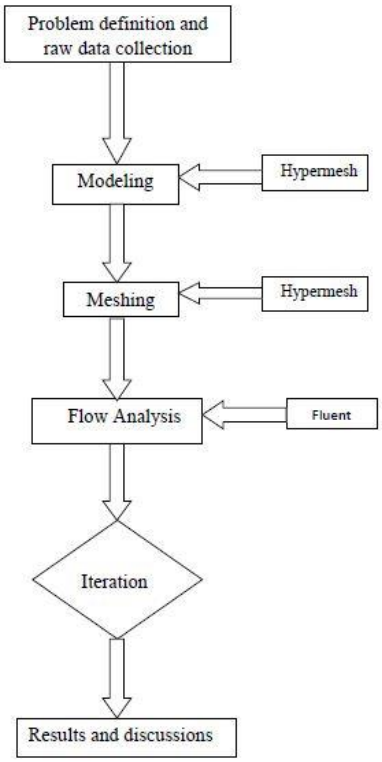


Fig-2: Flow Chart Representing the Methodology of Work

3. GOVERNING EQUATIONS

In this section we shall discuss about continuity equation, NS equations, Momentum Change and Flux and Turbulence K-Epsilon model.

4. MODELING

This section deals with the modeling of geometry and meshing of geometry. The modeling was done using Hypermesh (16.0) having the measurements of 1inch diameter, 5inch length [1-2] and for branching angle of 45, 60, 90 degrees. The schematic diagram Y- duct is shown in figure

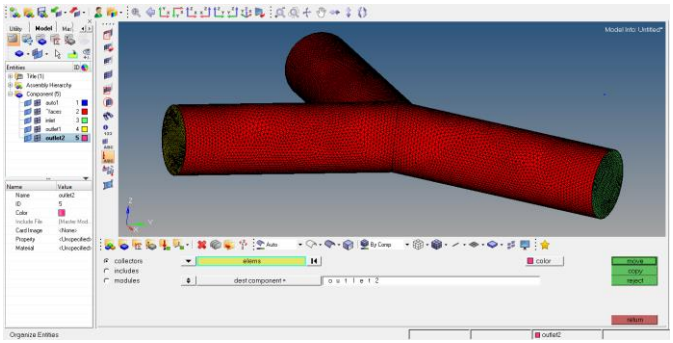


Fig-3: Modeling, Meshing and Boundry definition for 45^0

5. FLOW ANALYSIS

This section deals with the fluid flow simulation. The fluid flow simulation was done using Ansys Fluent (16.0) tool. Assuming that, the fluid flow through pipe as steady state, the viscous laminar model with K-epsilon governing equation was chosen to investigate the problem. The flowing fluid through the Y-Duct was selected as water surrounded with steel wall with their respective properties. The fluid entering the pipe as velocity and exiting as pressure outlet. The solution convergence was put under control by varying the parameters like momentum and turbulent kinetic energy. The problem was analyzed with the variation in velocity, ranging for 0.05 m/s.

6. RESULTS AND DISCUSSION

The velocity versus Pressure drop curve for different bent angle under consideration of velocity variation is illustrated in figure. In which, drop in pressure is high for an angle of bent 45^0 and it goes on reduce as the bent angle increases. This higher pressure drop was taken place in smaller bent angle due to high turbulence existence at junction. Also, pressure drop increasing continuously as there is an increase in velocity of fluid. It is observed that, the dynamic pressure developed at the inlet is high. This pressure development is due to the high velocity and high resistance offered by the contact surface on the flowing fluid. As, the fluid diverges into two path at the junction, the velocity decreases with the drop in dynamic pressure at the outlets.

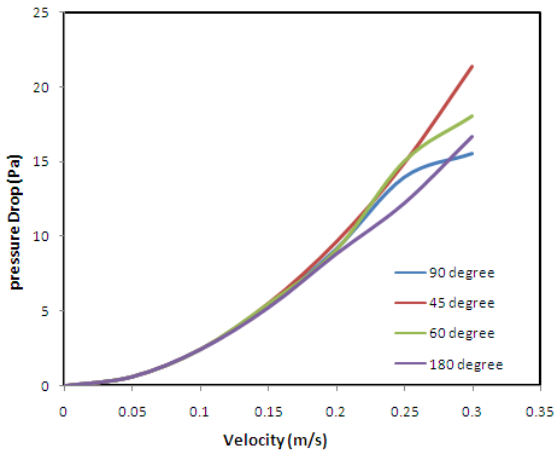


Fig-4: Pressure Drop versus Velocity

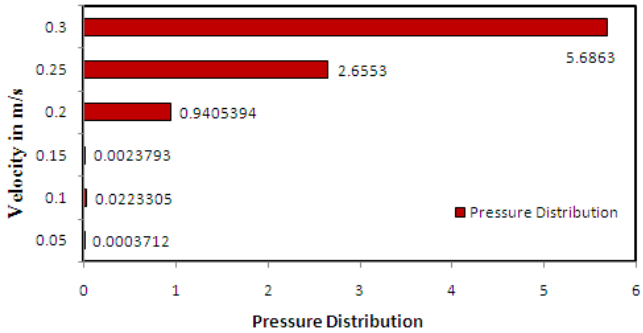


Fig-5: Pressure Distribution versus velocity

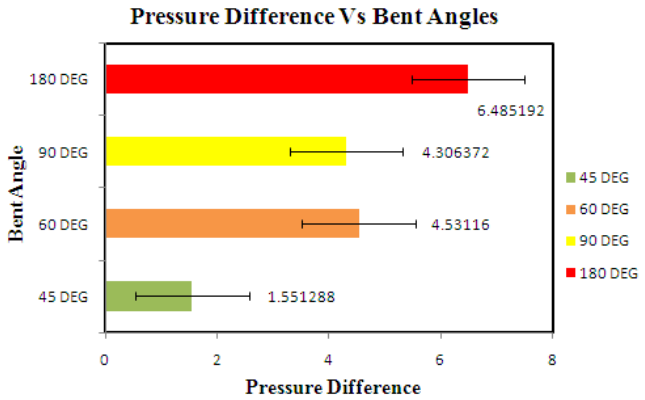


Fig-6: Pressure Distribution versus Bent angle

The velocity versus Pressure distribution at outlets is illustrated in figure (5). From this plot it can be conclude that, as the rate of velocity of flowing fluid increases, the asperous pressure distribution and uneven mass distribution occurs. The turbulence at the junction leads to cause asperous pressure distribution and uneven mass distribution because; the turbulence is the function of velocity and sudden interruption to flow.

The pressure distribution at outlet versus bent angle is illustrated in figure (6). Through this plot it was noticed that as the bent angle increases the difference in pressure distribution at outlets also increases. This unevenness may be due to the variation in the junction shape for different bent angles.

The difference between mass flow rates at the two outlets versus bent angle with respect to variation in rate of velocity is illustrated in figure (7). From this plot it was observed that for the unevenness in mass flow distribution at the junction is high for a branch of 1800.This unevenness occurrence in the mass flow distribution may be the result of sudden impact of fluid at junction and varying resistance on flowing fluid by the opposite surface.

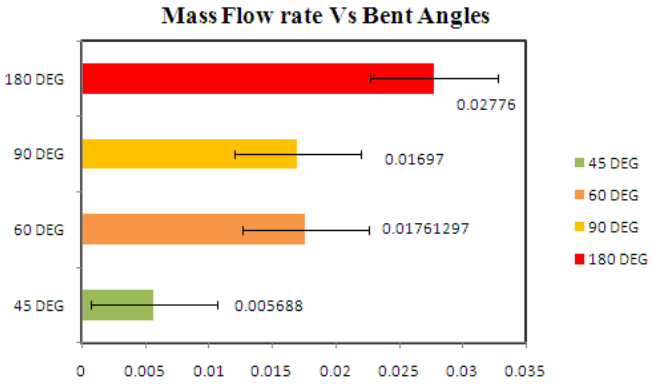
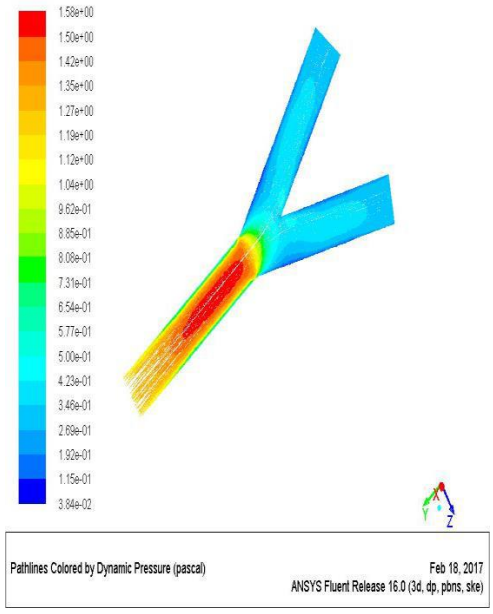


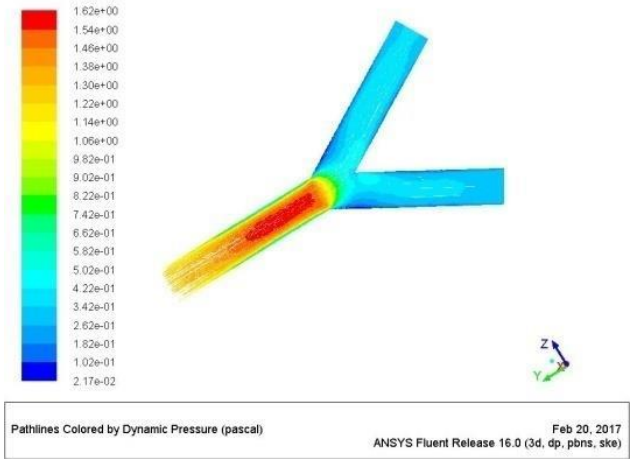
Fig-7: mass flow rate versus bent angle

- **Dynamic pressure plots for 0.05m/s Velocity Rate for different angles.**

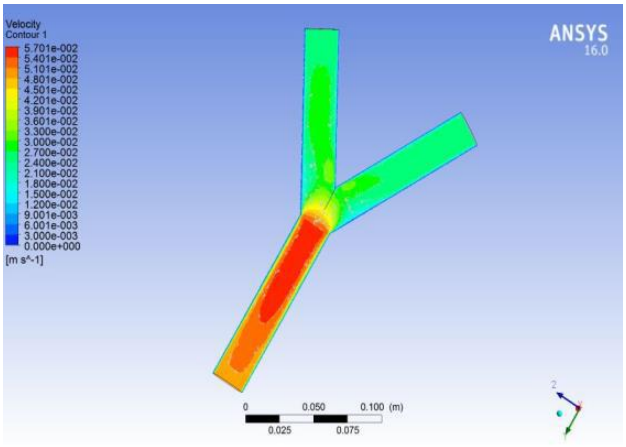
a. For 45⁰



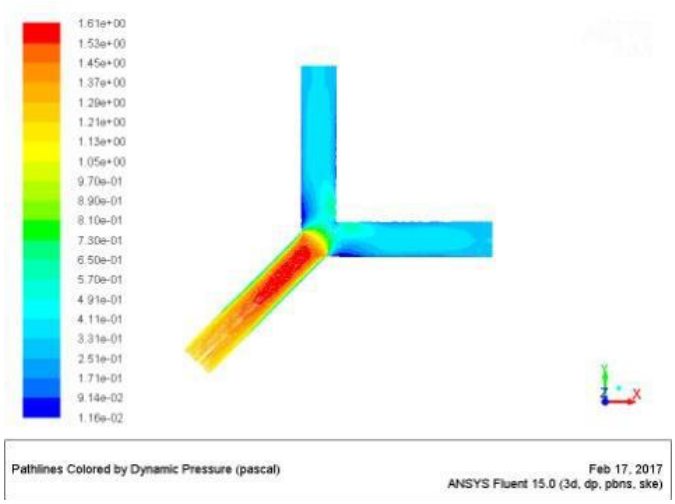
b. For 60⁰



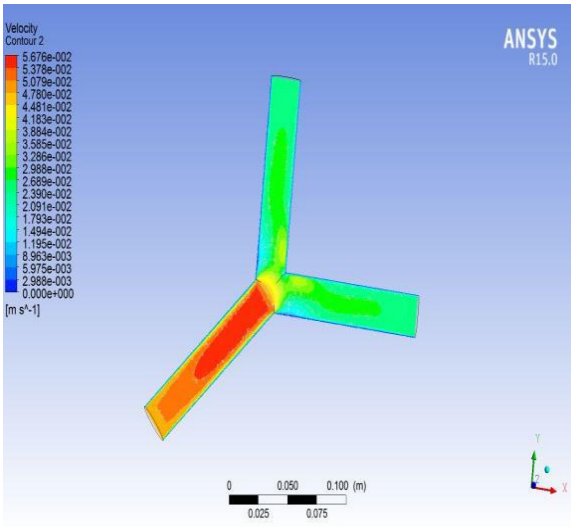
b. **For 60⁰**



c. For 90⁰

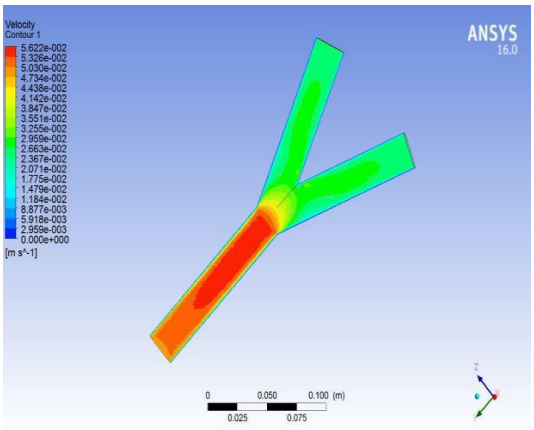


c. **For 90⁰**



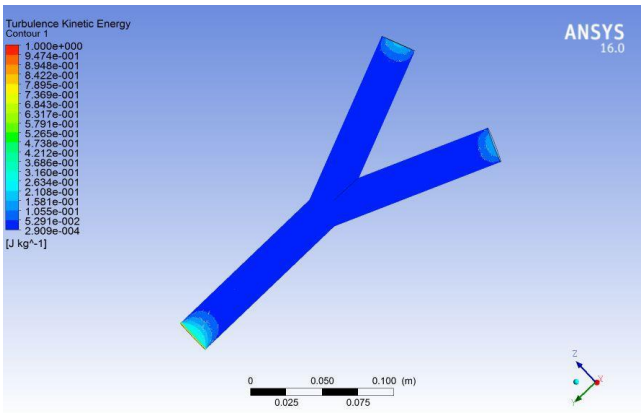
- **Velocity magnitude plots for 0.05m/s Velocity Rate for different angles.**

a. For 45⁰

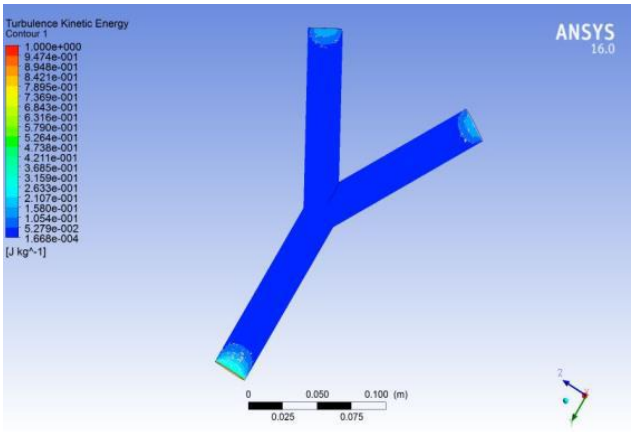


- **Turbulence kinetic energy (k) plots for 0.05m/s Velocity Rate for different angles.**

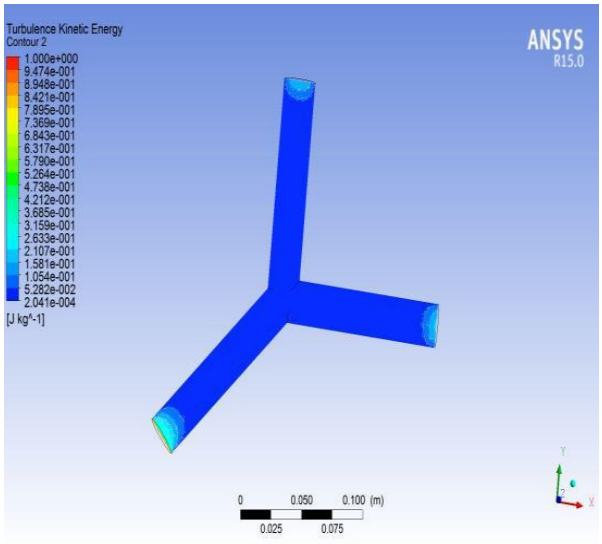
a. For 45⁰



b. 60^0



c. For 90^0



7. CONCLUSIONS

This section deals with the conclusion of this investigation:

- ✓ Through this investigation it has observed that, as the fluid passes through the junction, there subsist a drop in pressure and uneven flow distribution at the junction.
- ✓ This unevenness in flow distribution and pressure drop is the function of velocity and turbulence. Also, it has observed that the pressure distribution at the two outlets is uniform for lower rate of velocity.
- ✓ The less turbulence occurrence at 45º branching, results into unvarying pressure distribution at the outlets.
- ✓ The mass flow distribution at the branching zone is found to be uniform for the lower rate of velocities.
- ✓ This equality in rationing of flowing fluid is due to less angle segment and low turbulence occurrence at the distributing region. Thus, typically it can be wrap up that the loss in pressure, pressure distribution and mass flow distribution will be influenced by velocity parameter of flowing fluid.

REFERENCES

1. Gujarathi, Y. S. A Comprehensive Study on Numerical and Computational Aspects of Turbulence Modelling.
2. Hirani, A. A., & Kiran, C. U. (2013). CFD simulation and analysis of fluid flow parameters within a Y-Shaped Branched Pipe. IOSR Journal of Mechanical and Civil Engineering, 10 (1), 31-34.
3. Singh, B., Singh, H., & Sebgal, S. S. (2013). CFD analysis of fluid flow parameters within a Y-shaped branched pipe. International Journal of Latest Trends in Engineering and Technology (IJLTET), 2(2), 313-317.
4. Mr.G.B.Nimadge et.al, "CFD analysis of flow through T-junction of pipe", International Research Journal of Engineering and Technology (IRJET), Volume:4, e-ISSN: 2395 -0056 p-ISSN: 2395-0072, Feb-2017.
5. Dr. Neihad Al-Khalidy, "design optimization of industrial duct using computational fluid dynamics", Third International Conference on CFD in the mineral and Process Industries, CSIRO, Melborne, Australia, 10-12 December 2003
6. K.Venkateswara Rao, " Comparison of CFD simulation of hot and cold fluid mixing in t-pipe by placing nozzle at different places", International Journal of Research in Engineering and Technology, Volume: 03 Issue: 09, eISSN: 2319-1163 pISSN: 2321-7308, Sep-2014.
7. Mazumder, Q. H. (2012). CFD analysis of single and multiphase flow characteristics in elbow.
8. Anand, R. B., Singh, S. N., & Rai, L. (2008). Effect of swirl on the flow characteristics of S-shaped diffusing duct.