Velocity Effect Investigation on the Flow Parameters in Y-Duct

Vinayak S Uppin[1], Raghavendra N. Savannanavar[2], Vishwanath Paschapur [3]

[1] *Assistant Proffesor, Department of Mechanical Engineering, NNRG, Telangana, India*
[2] *Assistant Proffesor, Department of Mechanical Engineering, NNRG, Telangana, India*
[3] *Assistant Proffesor, Department of Mechanical Engineering, AGI, Karnataka, India*

ABSTRACT

The influence of velocity on the flow parameters had been investigated in this present work. The small junction of pipe network i.e. Y branch has 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. Ansys 16.0 i.e. Fluent CFD analysis tool was used to carry out the simulation part of existing problem of the fluid flow through Y-duct having 45º branching angle. The results are evaluated by varying the inlet boundary conditions i.e. for different velocities. Through this investigation it has observed that, as the fluid passes through the branch, there exist a drop in pressure and uneven flow distribution at the junction. The inequality in mass flow distribution and loss in pressure are the function of turbulence and velocity. Also, it has observed that the pressure distribution at the two outlets is uniform for lower rate of velocities. The less turbulence occurrence at 45º branching, results into unvarying pressure distribution at the outlets. Further, 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

Keyword: *Ansys16.0, Hypermesh 16.0, K-epsilon, Branch Angle, Velocity, Flow Distribution*.

1. INTRODUCTION

Pipe networks are being the transportation channels for the fluids since long ago. These systems vary from simple to complicated network. (e.g. petrochemical industry, steam power plants, water supply network). Generally, the piping networks will be consists of special components like Y-junction, elbows, gauges, valves, expansions, reducers etc. All these components tend to cause drop in pressure and uneven pressure distribution of flowing fluid due to change in momentum caused due to friction and pipe components. Thus, combinedly terminating into a loss of flow energy due to the turbulence and resistance offered by contact surface.

The investigated based on both numerical and analytical approach, considering the numerical models, governing equations, Reynolds equation, Navier Stokes equation and K-epsilon model for turbulence modeling states that the K-epsilon model can be used effectively to analyze the Y-duct problem, using Fluent 16.2 solver [1]. In pipe networking system, the angle of bend at branching has correlation with the coefficient of resistance offered by the adjacent surface on the flowing media. The coefficient of resistance to the flow will get increases with the increase of branching angle from 45º to 90º and its stand lower for an angle of 180º. This decrement is the result of sudden impact of fluid jet on the wall surface [2-3]. The analytical study made on T-joint concludes that, the junction will affect the flow parameters. Decrease in pressure and velocity occurs during the flow through T-joint. The pressure loss will be more when the fluid passes through T-segment, having angle of segmentation 90º and it can be decreases with variation in angle of segment. There will be a change in flow parameters as the fluid passes through T-junction [4]. An effort was made to optimize the flow parameters of flowing fluid through the duct by considering the industrial applications, which will result into small pressure drop, no re-circulation and low turbulence [5]. The investigation on mixing of two fluids concludes that, as the two fluids enters with differs velocities, it results in decrease of pressure of mixed flow. Also, the turbulence will be high at the intersection, because of sudden change in the cross section of pipe and velocity variation [6]. The profile of velocity and pressure at different locations indicate the increase in pressure at the elbow region and it gets decrease when the fluid passes the elbow. For both then single and double flow though elbow, the pressure behavior remains same [7]. The study made on the effect of various parameters like static pressure, total pressure, velocity and turbulence based on the mass average quantity reveals that for the swirl flow of 0.36 to 0.54 the pressure recovery coefficient increases with flow [8].

1.1 BRANCHING ANGLE AND ITS EFFECTS

In fluid flow analysis, modifying the configuration of geometry with respect to the branch angle plays an important role in pressure drop, distribution of flowing fluid and velocity fluctuation. The benefit of maintaining effective branch angle is to distribute the flowing fluid with proportion volume through branched segments. But, larger in angle variation may cause to build loss of pressure, velocity variation and turbulence. In general, branching may cause to form the complicate flow patterns. To know these effects lot of investigation have been carried out.

Thus, in this investigation an attempt is made to evaluate the affect of velocity variation of the flowing fluid on flow parameters like pressure drop, pressure distribution and mass distribution. For different velocities the parameters like Pressure drop, difference in pressure, difference in mass flow distribution are calculated and the suitable the parameters are recommended.

2. MODELING AND SIMULATION

The modeling was done using Hypermesh (16.0) having the measurements of 1inch diameter, 5inch length [2-3] and 45^0 branching angle. The schematic diagram and Geometry of Y- duct is shown in figure (1.a) and figure (1.b).

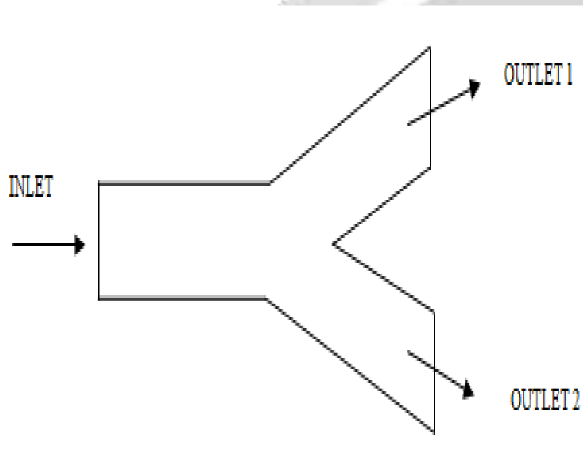


Fig-1.a Schematic Diagram of Y-duct

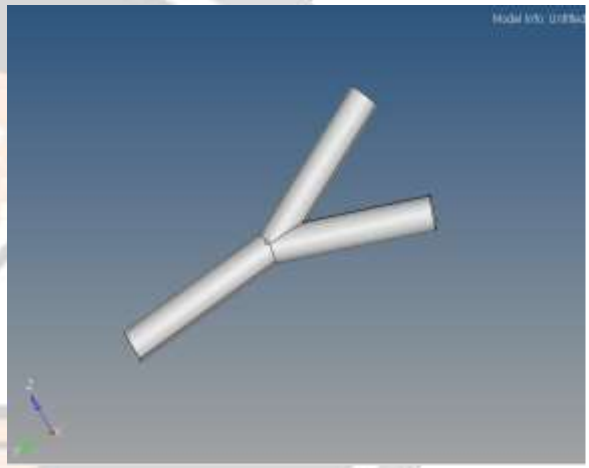
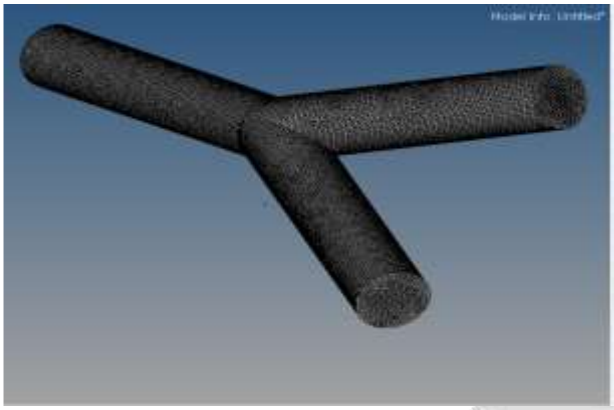


Fig-1.b Geometry of Y-duct

2.1 MESHING AND BOUNDARY DEFINITION

The model was meshed using Altair Hypermesh (16.0) tool; the model is descritised into small tetrahedron elements using volume tetra operation with the element size of 1.3. The meshing operation results the body into 256121 elements having nodes of 53270. The three different boundaries of the duct had been defined and assigned to the respective faces. The meshed and sliced Y-duct model is shown in figure (2.a) and figure (2.b).



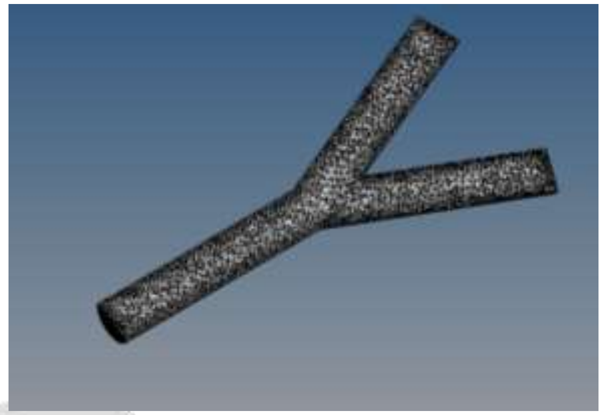


Fig-2.a. Meshed Model of Y-duct **Fig-2.b.** Sliced Model of Y-duct

2.2 SOLUTION STRATEGY AND CONVERGENCE

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 from 0.05-0.3 m/s with the incremental rate of .05m/s for each simulation.

3. RESULTS AND DISCUSSION

The velocity versus Pressure drop curve for velocity variation is illustrated in figure (3). In which, drop in pressure 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.

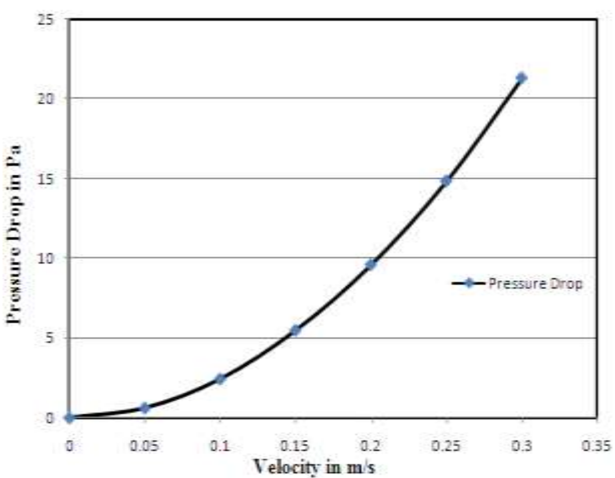


Fig-3 Pressure Drop versus Velocity

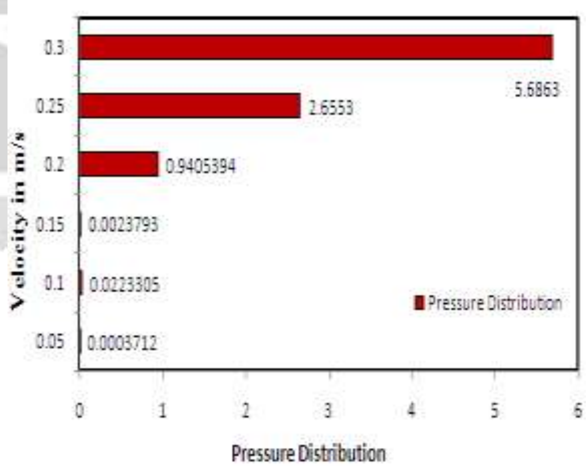


Fig-4. Pressure Distribution versus Velocity

The velocity versus Pressure distribution at outlets is illustrated in figure (4). 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 difference between mass flow rates at the two outlets with respect to rate of velocity is illustrated in figure (5).

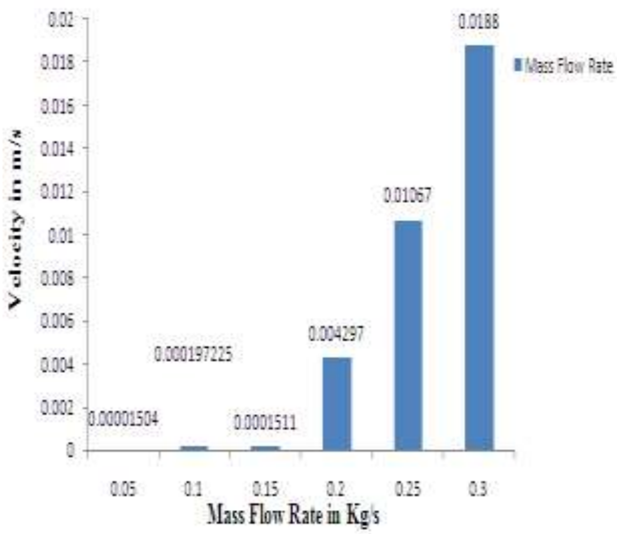


Fig-5. Rate of Flow versus Velocity

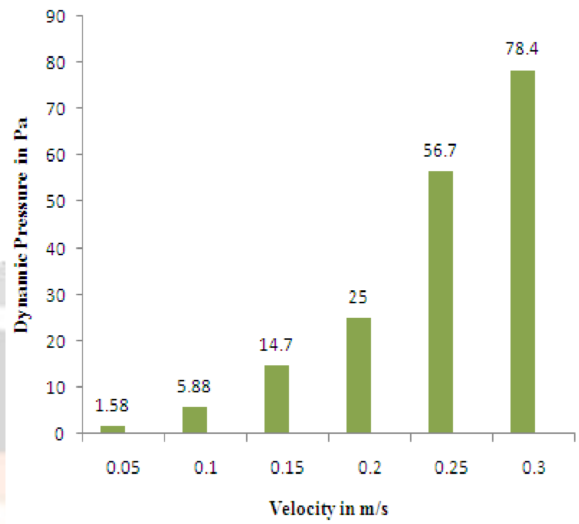


Fig-6. Pressure versus Velocity

The maximum dynamic pressure development with respect of velocity variation is adorned in figure (6).The gradual increase in the peak dynamic pressure can be observed in the plot with respect to the increase in velocity parameter. Co-efficient of resistance on flow and turbulence will influence this kind of behavior.

The simulation carried for different velocities picturize the velocity, pressure distribution and turbulence at the flow zone. Figure (7,8,9) illustrates the lower and critical velocity distribution, pressure distribution and turbulence existence.

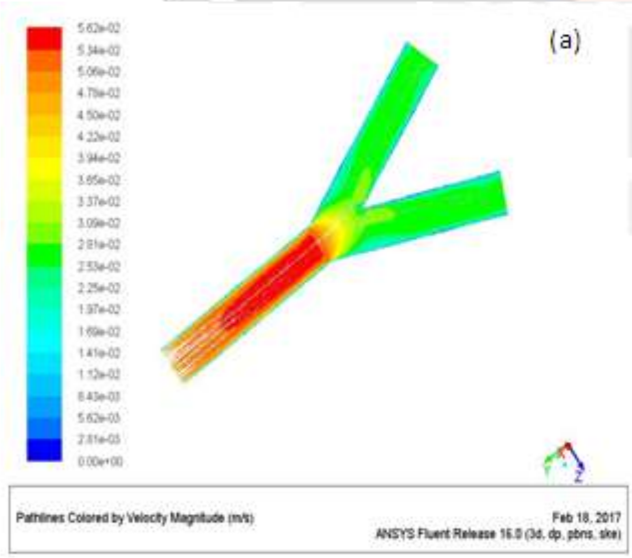


Fig-7.a Velocity Distribution having the rate of 0.05 m/s

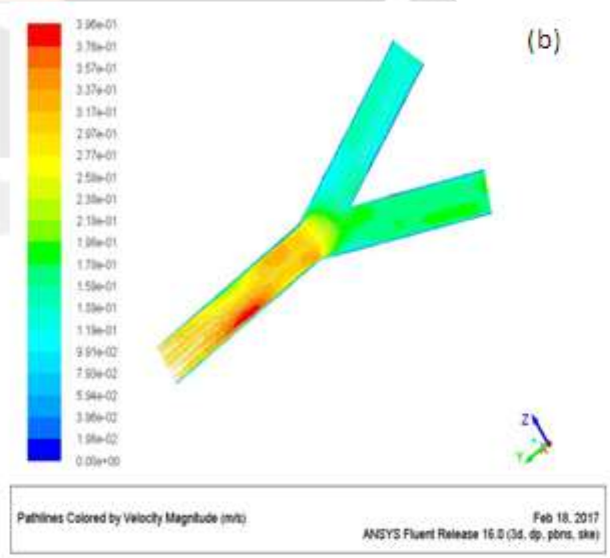


Fig-7.b Velocity Distribution having the rate of 0.3 m/s

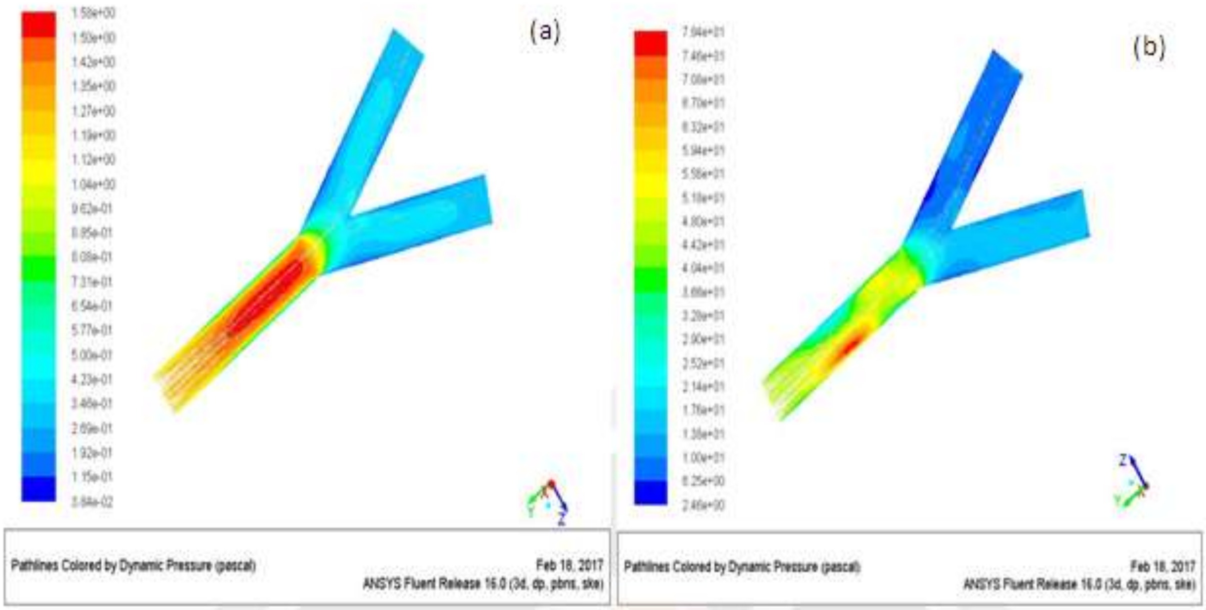


Fig-8.a Pressure Distribution for Velocity of 0.05 m/s **Fig-8.b** Pressure Distribution for Velocity of 0.3 m/s

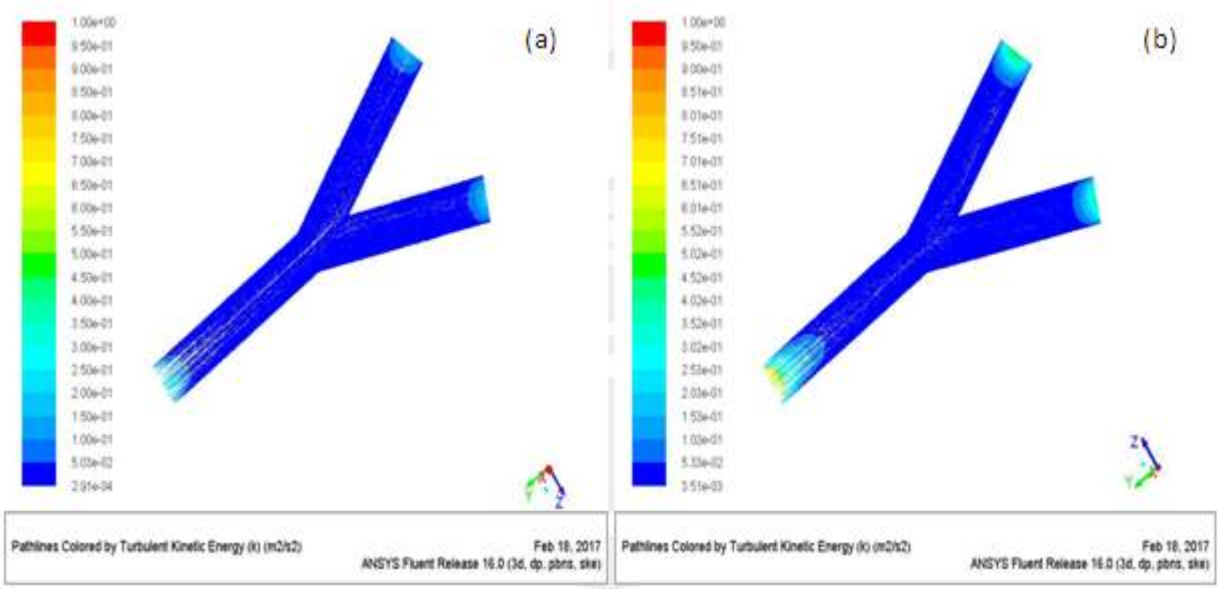


Fig-9.a Turbulence Existence for Velocity of 0.05 m/s **Fig-9.b** Turbulence Existence for Velocity of 0.3 m/s

4. CONCLUSIONS

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.

6. 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.