IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 10, Issue 1 (Nov. - Dec. 2013), PP 31-34
www.iosrjournals.org

CFD Simulation and Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe

Aslam A. Hirani[1], C. Udaya Kiran[2],

[1](*Department of Mechanical Engineering, J. B. Institute of Engineering and Technology, Yenkapally, Moinabad,, India)*

[2](*Department of Mechanical Engineering, J. B. Institute of Engineering and Technology, Yenkapally, Moinabad,, India)*

Abstract: *Plumbing system use pipe fittings to connect straight pipe or tubing section for regulating or measuring fluid flow. Y (wye)-shape fitting is one of the important component in the plumbing system. A wye branch allows splitting a branch line equally in two directions. The opening sizes can vary for different situations for instance in situation where a large main line needs to be split into two smaller branches. The wye shape fitting will convert into T shape fitting when the included angle between two pipe branches is 180°. In the present work, effect of angle of turn/bend for a Y-shape pipe will be studied computationally using ANSYS CFX software. For the analysis, all the three pipe branches of 1 inch internal diameter are selected along with equal length so that only the effect of bend angle at* 45^0, 60^0, 90^0 *and* 180^0 *can be studied. Water as a fluid is selected which flows through the plumbing system. The effect of bend angle, pipe diameter, pipe length, Reynolds number on the resistance coefficient is studied. It was observed that resistance coefficient vary with the change in flow*

Keywords: *Y (wye)-shape, effect of bend angle, pipe diameter, resistance coefficient, ANSYS CFX.*

I. INTRODUCTION

Pipe networks are very common in industries, where fluid or gases to be transported from one location to the other. The pressure loss may vary depending on the type of components coming across in the network, material of the pipe, the fluid that is being transported through the network and pipe fitting. The analysis of pipe network is very important in engineering point of view. A lot of engineering problem dealt with it. Due to rigorous engineering application and implications the analysis is important. Fittings are used in pipe and plumbing systems to connect straight pipe or tubing sections, to adapt to different sizes or shapes, and for other purposes, such as regulating or measuring fluid flow. A wye branch (as the name implies, this fitting device is "Y" shaped) allows splitting a branch line equally in two directions. It is a fitting with three openings and is used to create branch lines [2]. A standard wye allows one pipe to be joined to another at a 45 degree angle. Wyes are similar to this except that the branch line is angled to reduce friction that could hamper the flow and that the connection is typically at a 45-degree angle rather than a 90-degree angle [1]. Economy wyes are often spot welded together; industrial wyes have a continuous weld at each seam. In the present work, fluid dynamic analysis for different Y- shape connections will be done, for laminar. Effect of angle of turn/bend for a Yshape pipe for different Reynolds number will be studied.

II. PROBLEM SPECIFICATION

Figure (1) shows a schematic representation of the flow distribution through pipe and a general physical setup. Fluid enters the pipe at one end and exit from the two pipes at other end placed at different bend angle. To analyse the fundamental system properties and flow patterns, a simplified flow model was employed in this study [3]. All the three pipe branches of 1 inch internal diameter are selected along with equal length so that only the effect of bend angle at 45^0, 60^0, 90^0 and 180^0 can be studied [4]. The area ratio between the main pipe and branch pipe considered as A1/A2=1. Water as a fluid is selected which flows through the plumbing system. Water enters at a uniform temperature at T = 25 ^{0}C and at constant velocity of 0.05 m/s^2 at inlet. In this work, we are investigating the following effect of bend angle, pipe diameter, pipe length, Reynolds number on the resistance coefficient.

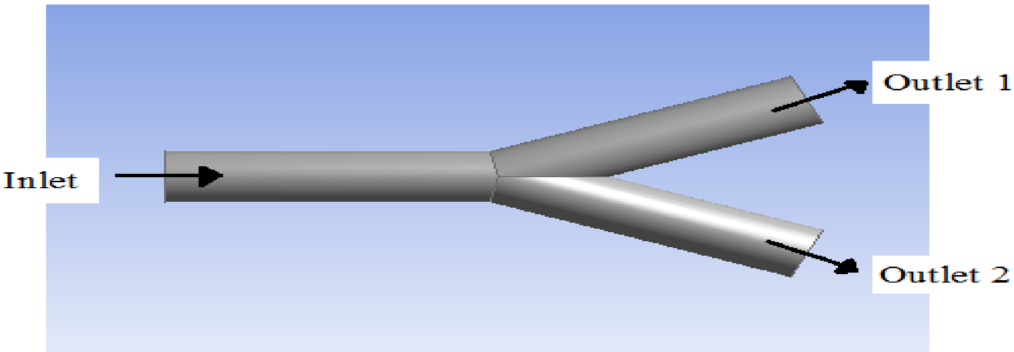

Fig. 1 Schematic representation of the flow distribution through Y shape branch

III. MODELING AND SIMULATION OF Y SHAPE BRANCH IN ANSYS CFX

The Y shape branch in modelled in ANSYS CFX. Diameter of each pipe is 1 inch and have equal length. Inlet pipe is horizontal and outlet pipes are at angle of 45^0, 60^0, 90^0 and 180^0 are modelled [7]. Meshing is done using tetrahedrons element which in as shown in figure 2. Total number of tetrahedrons element generated are 18179 [5]. The fluid used in the simulation is water with constant density 997 kg/m^3 and dynamic viscosity 0.0008 kg/ms. The fluid is assumed to be incompressible. The boundary condition were set as mass flow at inlet and pressure at two outlets. Fluid enter at a uniform temperature at T = 25 ^{0}C and at constant velocity of 0.05 m/s^2 at inlet and outlet condition are NTP [6].

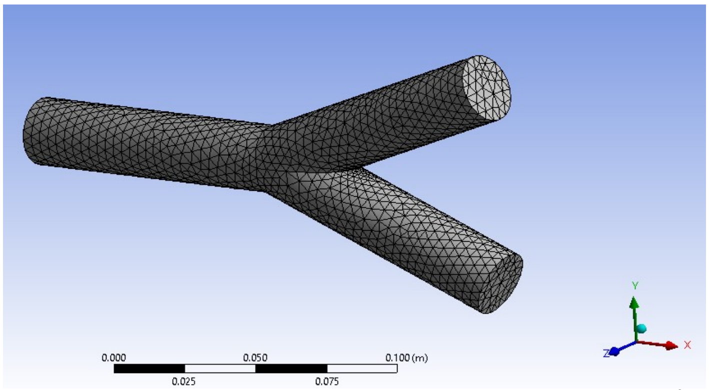

Fig. 2 Meshing of Y shaped branch at an angle of 45^0

IV. CFD RESULT AND DISCUSSION

The CFD analysis for the Y shape pipe joint was done using ANSYS CFX, for four angles 45^0, 60°, 90° and 180° was selected in the current work. The angle 180°converts the wye joint into T- joint. Figure 3 shows the velocity profiles for Y-Joint with angle (θ=45°), inlet velocity of 0.05 m/s and outlet condition as NTP.

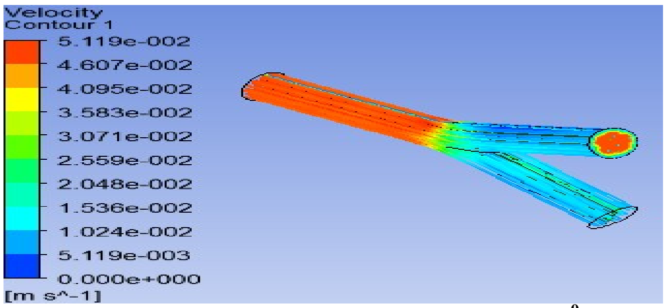

Fig. 3 Velocity profile at an angle of 45^0

Figure 4 shows the velocity profiles for Y-Joint with angle (θ=60°), inlet velocity of 0.05 m/s and outlet condition as NTP.

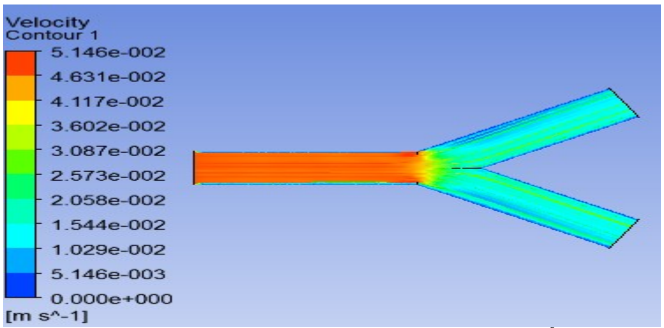

Fig. 4 Velocity profile at an angle of 60^0

Figure 5 shows the velocity profiles for Y-Joint with angle (θ=90°), inlet velocity of 0.05 m/s and outlet condition as NTP.

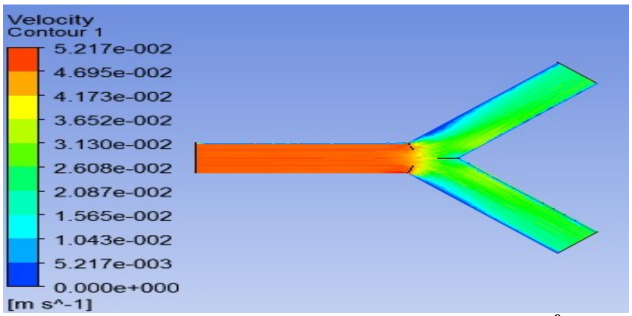

Fig. 5 Velocity profile at an angle of 90^0

Figure 6 shows the velocity profiles for Y-Joint with angle (θ=180°), inlet velocity of 0.05 m/s and outlet condition as NTP

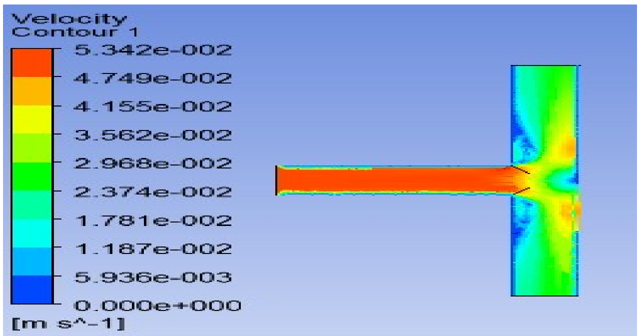

Fig. 6 Velocity profile at an angle of 180^0

Table 1 shows the variation of resistance coefficient for water flow at different bend angle. It was observed that as the bend angle increases from 45 to 90 the resistance coefficient also increases, but at angle of 180 it again decreases the reason for the same is attributed due to sudden impact of the water jet on the pipe wall and flow distributes without much resistance.

TABLE I
VARIATION OF RESISTANCE COEFFICIENT AT DIFFERENT BENT ANGLE FOR WATER FLOW

Bend angle	θ=45°	θ=60°	θ=90°	θ=180°
Resistance coefficient	0.016	0.016	0.016	0.015
Pressure drop	0.01	0.01	0.01	0

V. Conclusion

From the CFD analysis it was analyzed that the secondary flow caused by a fully developed straight pipe profile at the start of the bend increases to a maximum at a bend angle of about 60°. From this point onwards, the total secondary flow is reduced until by about 90° it reaches a steady value. In the long bend, a mechanism for preventing the secondary flow increasing indefinitely, and for allowing the flow to become fully developed, is provided by the formation of total pressure gradients, opposite in sign to those at the start of the bend, and the consequent production of vorticity of opposite rotational sense.When the bend is increases beyond 90° so that this negative vorticity becomes positive, the secondary flow tends to increase. An important outcome of the CFD analysis which validates the practical application of wye pipe at bend angle of 45° was the resistance coefficient which comes out to be zero

REFERENCES

[1] Romero-Gomez, P., C. K. Ho, and C. Y. Choi. (2008). Mixing at Cross Junctions in Water Distribution Systems – Part I. A Numerical Study, *ASCE Journal of Water Resources Planning and Management,* 134:3, pp. 284-294.

[2] Austin, R. G., B. van B. Wanders, S. McKenna and C. Y. Choi. (2008) Mixing at Cross Junctions in Water Distribution Systems – Part II. An Experimental Study, *ASCE Journal of Water Resources Planning and Management* 134:3 pp. 295-302.

[3] Ho, C. K. Solute Mixing Models for Water Distribution Pipe Networks. *ASCE Journal of Hydraulic Engineering*. Vol. 33, Issue 3, pp.830-836, 2008

[4] Anand, R.B., Sandeep, Reflect of angle of turn on flow characteristics of Y-shaped diffusing duct using CFD, *Frontiers in Automobile and Mechanical Engineering (FAME)*, 2010, 25-27 Nov. 2010.

[5] R.B. Anand, A. Chandraprabhu, X.J.A. Richards, N. Hares ram, Flow and performance characteristics of a Y-shaped diffusing duct using CFD*, International Journal of Aerodynamics* 2010 - Vol. 1, No.2 pp. 115 - 129.

[6] B.J.Mc Keon, M.V.Zagarola, A.J.Smits, A new friction factor relationship for fully developed pipe flow, J. Fluid Mech. (2005), vol. 538, pp. 429–443

[7] Gyorgy, Pinho, and Maia, 2006. "The effect of corner radius on the energy loss in 90˚tee junction turbulent flows". Department of civil engineering, Faculty of Engineering, University of Porto