International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

STUDY OF VELOCITY DISTRIBUTION ACROSS PIPE TRIFURCATION USING COMPUTATIONAL FLUID DYNAMICS

Syed mustafkhadri[1], Dr. Kishan naik[2], Mr. Kotresh banjara[3]
Asst. Professor, asst.professor, asst.professor
Department of mechanical engineering
Giet[1], ubdtce[2], govt. Engineering college[3]
Moinabad, davanagere, haveri, india
smustafa.k22@gmail.com,kishennaik@gmail.com,bkotresha@gmail.com

---

Abstract

A trifurcation represents a main pipe branching out or dividing into three branches. It divides the flow from main pipe to branched pipes. At the centre of trifurcation there is a negative energy loss occurs. The losses inside trifurcation are represented by the range of vortex zone generation. Due to these losses, variation in velocity takes place. Hence CFD analysis of velocity distribution across the pipe trifurcation has very much importance. Computational investigation of turbulent flow with different Reynolds Numbers inside a pipe trifurcation is presented in this project. Pipe trifurcation geometry was created and mesh was generated in ANSYS GAMBIT and ANSYS FLUENT 15 software is used to implement this three-dimensional CFD model. The fluid used for this purpose is water and angle of trifurcation is 10^{o}. Inlet velocities are 3, 4 and 5 m/s respectively. The variation of velocity magnitude along the length of pipe is analysed and mass flow rate is also obtained.

Index Terms—Trifurcation, ANSYS, CFD, GAMBIT, FLUENT 15, Mesh, Velocity magnitude, Reynolds number, Vortex, Mass flow rate.

I. INTRODUCTION

In this world there are various types of applications of pipe flow, such as plumbing, tap water, in petroleum industry for transportation of liquids or gases over long distances using pipe lines etc. Pipes are hollow cylinders used to convey flowing fluids (liquids and gases) and any chemically stable substances from one location to another. Various types of material such as glass, fibre glass, plastic, concrete, ceramic and many metals such as steel or iron are used for making pipes. Pipe lines are useful for transportation of water for drinking and irrigation purpose over long distances where canals or channels are not a good choice due to effect of evaporation, pollution and impact of environment. Steel or plastic pipes are used for oil pipelines which are generally buried. The movement of oil through pipelines can be established by pump stations. Carbon Steel is used for construction of pipeline to transport Natural gas. We use a network of pipes with insulation to transport hot water, pressurized hot water or steam. While transporting explosive or flammable material we should follow safety methods to avoid accidents. The trifurcations are elements of frame work that establishes the hydro electric plant, together with other elements, parts and equipments to generate electricity utilizing the potential of hydraulics present in a dam or river. To get better operating condition it is essential to minimize the losses.

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

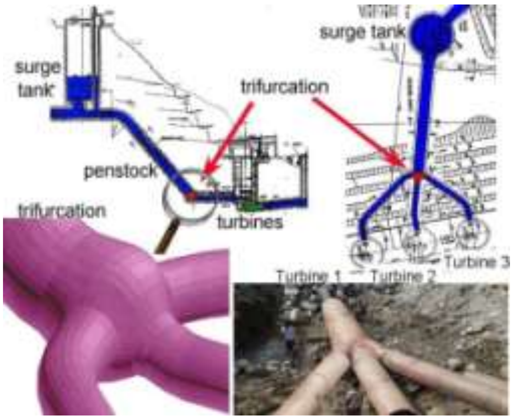

Fig. 1. Pipe trifurcation

II. BACKGROUND INFORMATION

a. Governing equations

The governing equations for the fluid flow exhibit numerical explanation of the conservation laws of physics. These equations are responsible for the motion of the fluid. In vector form, we can write the three basic governing equations as:

Continuity or mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \mathbf{V}) = 0 \tag{1}$$

Momentum equation:

$$\rho \frac{d\boldsymbol{V}}{dt} = \rho \boldsymbol{g} - \nabla.\boldsymbol{p} + \nabla.\tau_{ij} \tag{2}$$

Energy equation:

$$\rho \frac{\partial E}{\partial t} = \nabla.q - \boldsymbol{p}(\nabla.\boldsymbol{V}) + \emptyset \tag{3}$$

Where ρ is fluid density, $\boldsymbol{V}$ is fluid velocity vector, $\boldsymbol{g}$ is the body forces due to gravity, $\boldsymbol{p}$ is pressure, τ_{ij} is viscous stress tensor, t is time, E is internal energy, q is heat lost by conduction, φ is dissipation term.

b. Separation of flow

The separation of flow takes place when the velocity at the wall is zero or negative and formation of point of inflection in velocity profile and when a positive or adverse pressure gradient takes place in flow direction.

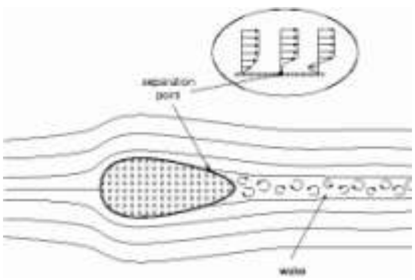

Fig. 2. Separation of flow

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

c. Loss in energy

Energy at inlet = Energy at outlet + Loss in energy
Loss in energy = Energy at outlet – Energy at inlet
Energy at inlet per unit time = Work done by pressure per unit time + Kinetic energy

$$= PQ + \frac{\rho\, QV^2}{2} \tag{4}$$

Energy at outlet / time $= \frac{\rho Q_1 U_1^2}{2} + \frac{\rho Q_2 U_2^2}{2} + \frac{\rho Q_3 U_3^2}{2}$ (5)

Where P: Static pressure (Pascal), ρ : Density of water (Kg/m³) , Q: Discharge in main pipe (m³/s) , Q_1, Q_2, Q_3: Discharges in branching (m³/s) , V : Velocity in main pipe (m/s) and U_1, U_2, U_3: Velocities in branching (m/s).

d. Boundary conditions

The boundary conditions used for analysis are as follows:

- At entry of the pipe trifurcation ----Velocity inlet
  Velocity magnitudes (m/s) = 3, 4, 5 (for water)
- At the walls ---- no slip condition
- At exit-----Pressure outflow p = 0 Pa

The no-slip condition is a proper predicament for the velocity ingredient at solid wall. The velocity is fixed to zero at the boundary.

e. Specifications:

Table 2 Pipe Trifurcation Dimensions [5]

Parameters	*Values*
Diameter of the main pipe (m)	*0.0254*
Length of the main pipe (m)	*1.37*
Diameter of branched pipes (m)	*0.0196*
Length of branched pipes (m)	*0.762*
Angle of trifurcation	*10°*

Properties of fluid (water) and other parameters [4]

- Type of flowing fluid Turbulent
- Flowing fluid water
- Temperature for both the fluids (K) 283
- Density of the water (Kg/m^3) 999.73
- Kinematic viscosity of the water (m²/s) $1.307 * 10^{-6}$
- Material of the pipe Steel

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

f. Determination of nature of flow

- Reynolds Number

$$R_e = \frac{VD}{\vartheta} \qquad (6)$$

Where
V – Inlet Velocity of the fluid in main pipe
D – Diameter of the main pipe
ϑ – Kinematic viscosity of water

g. Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) is the analysis of assemblies consisting fluid flow, heat transfer and related chemical reactions through computer based simulation. It uses numerical analysis and algorithms for solving and analyzing the problem that involves fluid flows. CFD codes consists three main elements (a) Pre-processor, (b) Solver and (c) Post-processor.

h. Numerical resolution

The economical CFD software FLUENT 15 is used to simulate the fluid flow and other fields. Grid is generated for the solver by using GAMBIT of the same package.

- Geometric characterization and grid generation
- Boundary zones
- FLUENT 15 set up
- Characterization of models
- Assigning the material features
- Defining the boundary conditions
- Case initialization and execution of FLUENT 15 code or run the calculations

III. COMPUTATIONAL DOMAIN

a. Geometry and model

The geometric performance is carried out in three-dimensional domain which manifests a pipe trifurcation through which fluid (water) flows. The geometrical domain used in this work is presented in Figure 3.

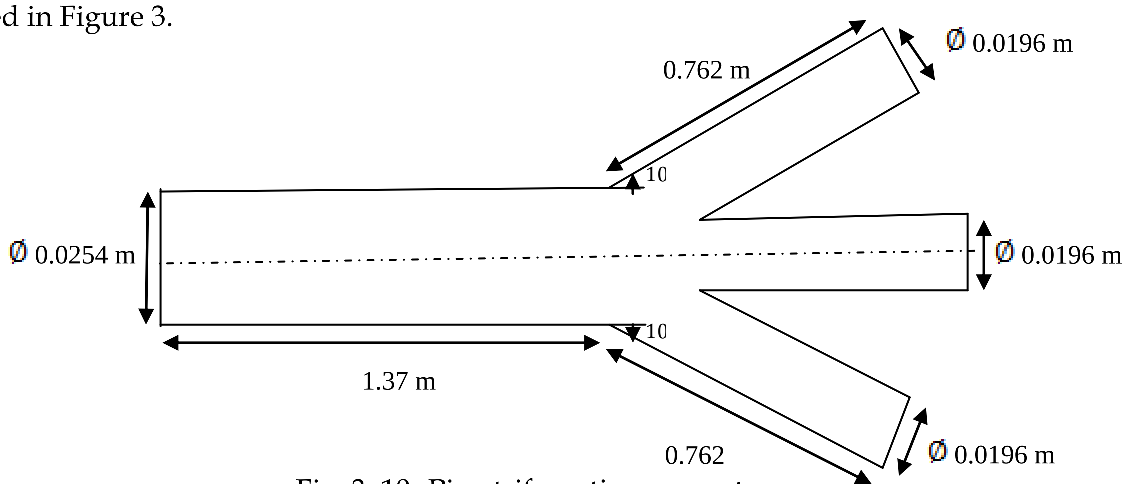

Fig. 3. 10° Pipe trifurcation geometry

A pipe trifurcation three-dimensional model created using ANSYS GAMBIT is shown in Figure 4.

Fig. 4. Three-dimensional model of 10° pipe trifurcation

b. Meshing

Meshing of the model is brought about to raise the accurateness of the model. Mesh or grid is well-described as a different cell or elements into which the domain is separated. The flow parameters and any other variables are analyzed at centre points of these individual elements. The important types of mesh/grid used are: Tetra meshing, hexagonal meshing, prism meshing and hybrid meshing. The details of mesh applied for the 10° pipe trifurcation is tabulated in the Table 1.

Table 1 Mesh details for 10° pipe trifurcation

Element	*Type*	*Mesh volumes*
Tet/hybrid	*T Grid*	*189602*

The meshed pipe trifurcation is shown in Figure 5.

Fig. 5. Mesh generation for 10° pipe trifurcation

iJCEM
International Journal of Core Engineering & Management

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

IV. RESULTS

Case 1) For Inlet velocity = 3 m/s

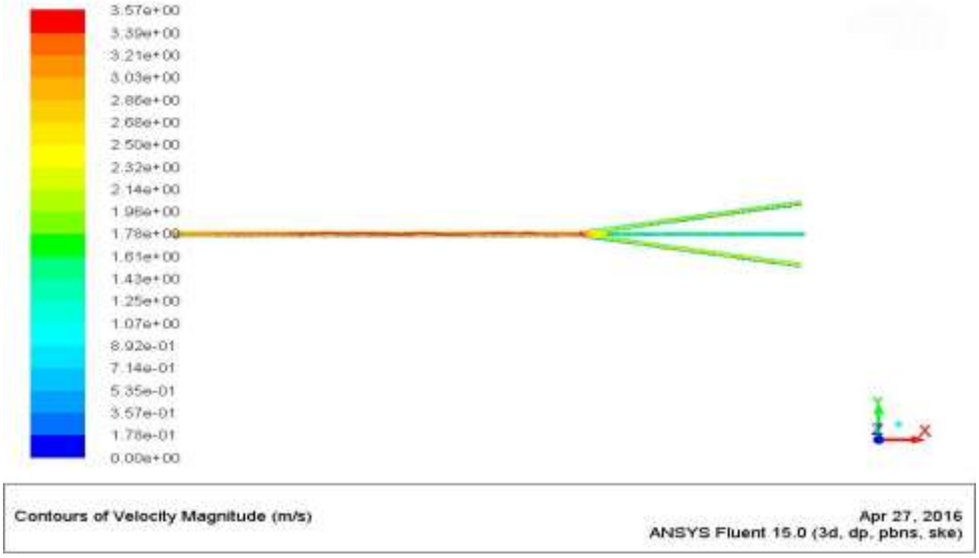

Fig. 6. Velocity distribution for inlet velocity 3 m/s

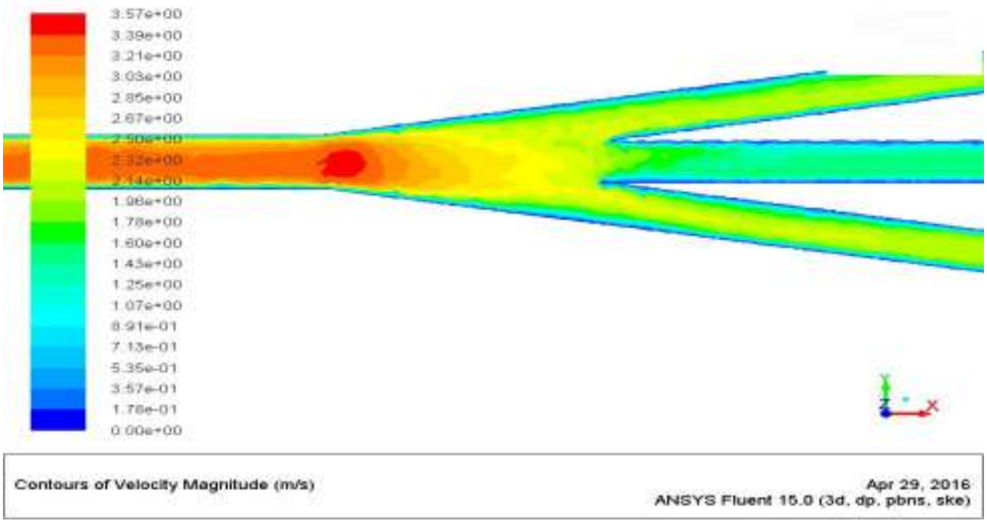

Fig. 7. Zoomed view of velocity distribution for inlet velocity 3 m/s

Velocity distribution contour plots across pipe trifurcation for inlet velocity 3 m/s are shown in Figures 6 and 7 respectively. As read from the above figures, starting from inlet, velocity is maximum along the centre axis, goes on decreasing as we move away from it and reaches to zero near walls. The velocity is equal to zero at the walls because of no slip boundary condition at the walls. It is also observed that the large vortex formation at the end of main pipe and entry of trifurcation zone which leads to decrease in pressure. It is the effect of flow separation and generation of back pressure. After that, the velocity is slightly decreased in the left and right branched pipes whereas it is decreased too much in the centre branch due to flow separation and branching of pipe.

Case 2) For Inlet velocity = 4 m/s

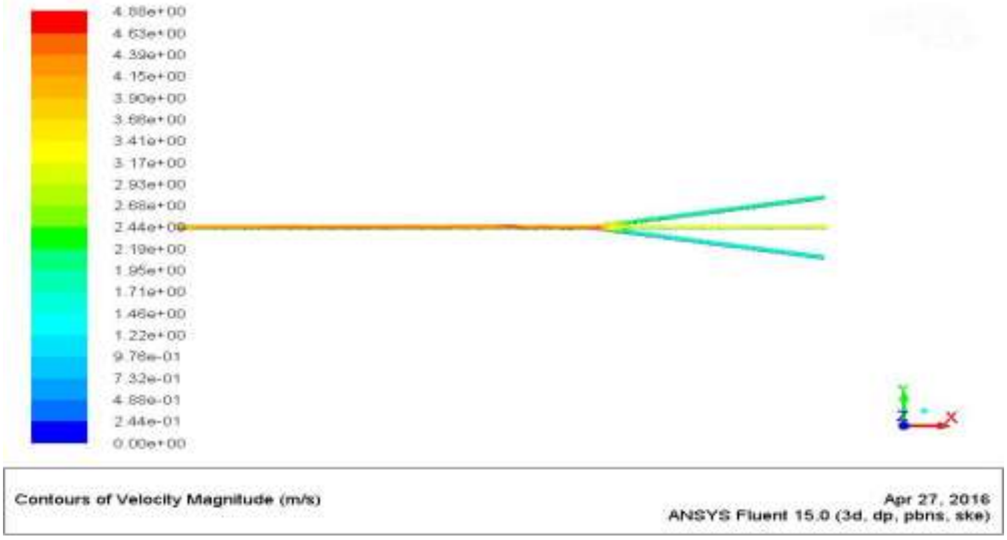

Fig. 8. Velocity distribution for inlet velocity 4 m/s

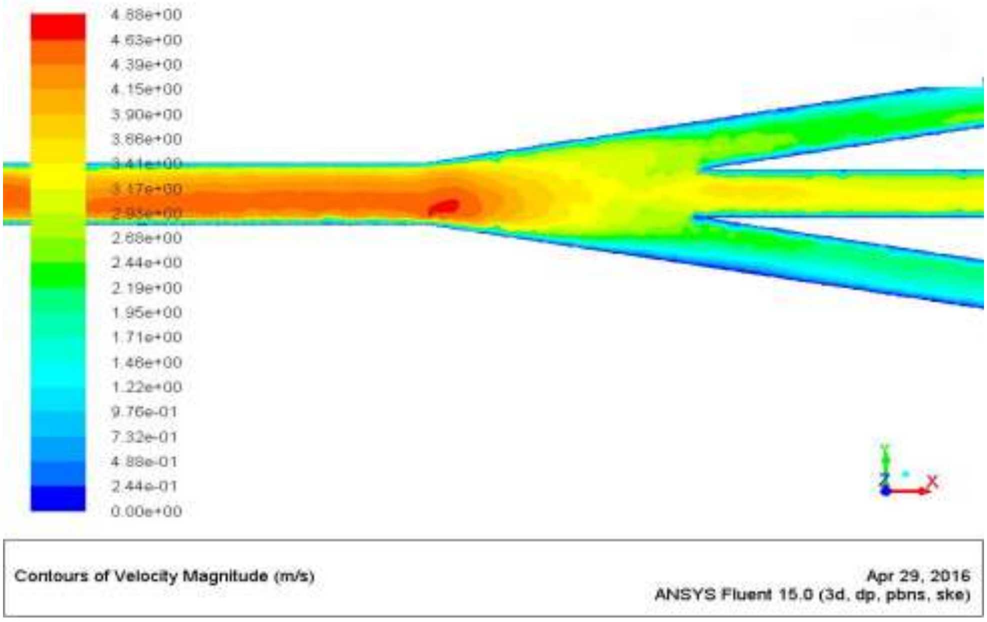

Fig. 9. Zoomed view of Velocity for inlet velocity 4 m/s

Velocity distribution contour plots across pipe trifurcation for inlet velocity 4 m/s are exposed in Figure 8 and Figure 9 respectively. It is contemplated from above figures that, starting from inlet the velocity is maximum along the centre axis, goes on decreasing as we move away from it and reaches to zero near walls. The velocity is equal to zero at the walls because of no slip boundary specification at the walls. It is also observed the vortex formation at the end of main pipe and entry of trifurcation zone which leads to decrease in pressure. It is the effect of flow separation and generation of back pressure. After that, the velocity is decreased to a lower value in the left and right branched pipes compared to the centre branch due to flow separation and branching of pipe.

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

Case 3) For Inlet velocity = 5 m/s

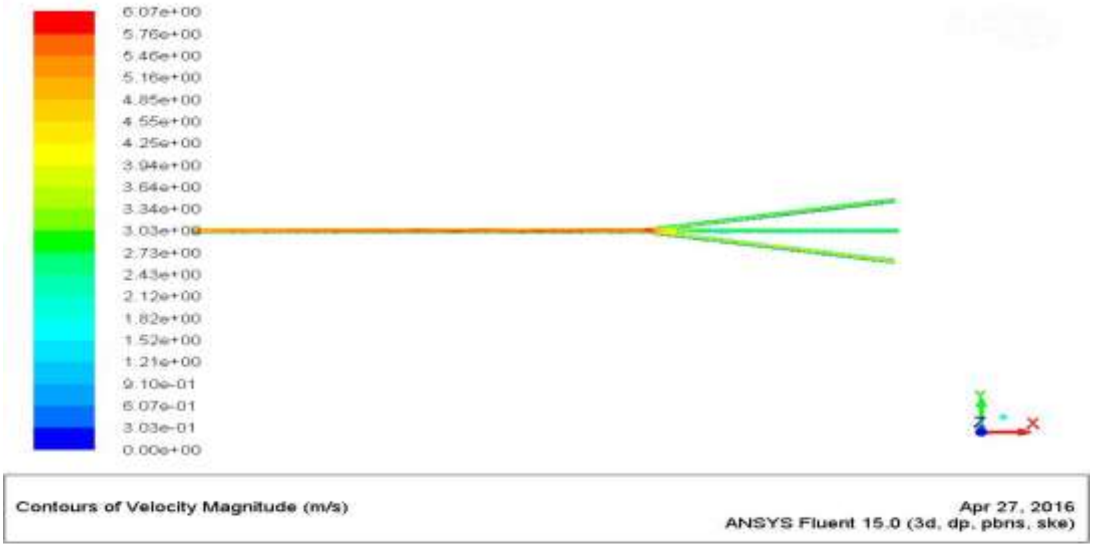

Fig.10. Velocity distribution across for inlet velocity 5 m/s

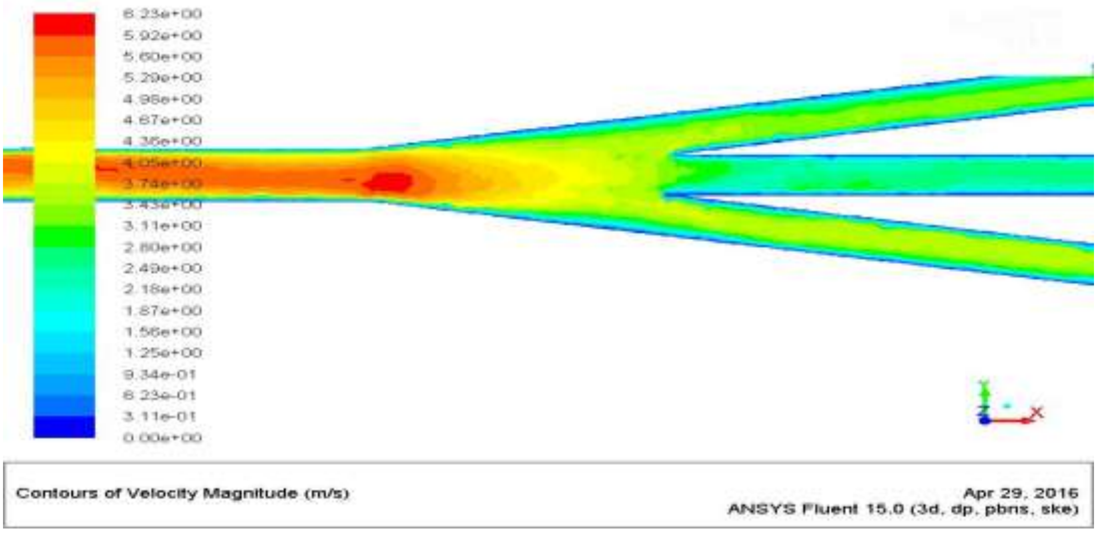

Fig. 11. Zoomed view of Velocity distribution for inlet velocity 5 m/s

Velocity distribution contour plots across pipe trifurcation for inlet velocity 5 m/s are displayed in Figure 10 and Figure 11 respectively. Starting from inlet, the velocity is maximum along the centre axis, goes on decreasing as we move away from it and reaches to zero near walls. The velocity at the walls is equal to zero because of no slip boundary at the walls. It is also observed the very large vortex formation at the end of main pipe and entry of trifurcation zone which leads to decrease in pressure. It is the effect of flow separation and generation of back pressure. After that, the velocity is decreased in the left and right branched pipes whereas it is decreased too much in the centre branch due to flow separation and branching of pipe.

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

V. COMPARISON OF RESULTS

a. Comparison of Velocity variation across the pipe trifurcation for different inlet velocities

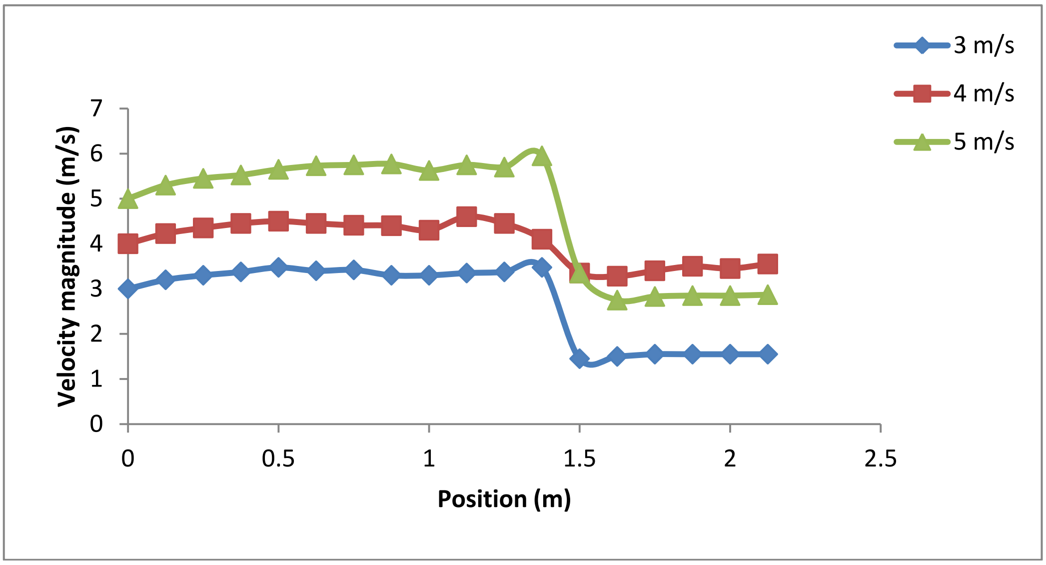

Fig. 12. Variation of velocity across pipe trifurcation

Figure 12 reveals velocity distribution for turbulent flow of water through pipe trifurcation. Beginning from entry of the main pipe velocity changes slightly up to trifurcation zone and then it will increase to a higher value and falls to some lower value in that zone due to distribution of velocity in branches. After the zone, it will change slightly as in the case before the trifurcation zone. As it is observed, drop in the velocity is more in the case where inlet velocity is 3 m/s compared to the cases where inlet velocity is 4 and 5 m/s respectively.

b. Comparison of calculated and ansys results of mass flow rate for different inlet velocities

Table 3. Comparison of Calculated and ANSYS results

Sl. No	*Inlet Velocity (m/s)*	*Mass Flow Rate (Kg/s) in main pipe*	
		Calculated	*ANSYS*
1	*3*	*1.5216*	*1.5001122*
2	*4*	*2.0288*	*2.0014950*
5	*5*	*2.5361*	*2.5001869*

VI. CONCLUSION

Main conclusions resulting from this work are as follows: Vortex formation increases with increased turbulence. High scale vortex formed for inlet velocity = 5 m/s. Starting from the entry of the main pipe velocity changes slightly up to trifurcation zone and then it will increase to a higher value and falls to some lower value in trifurcation zone due to distribution of velocity in branches. Drop in the velocity is more in the case where the inlet velocity is 3 m/s compared to cases where inlet velocity is 4 and 5 m/s respectively. ANSYS results of mass flow rate for different inlet velocities are well agreement with calculated results. Loss in energy is increasing with increased inlet velocities due to increased disturbances. There is greater loss in energy for inlet velocity 5 m/s.

REFERENCES

[1] R.K Malik, Paras Paudel, "3D Flow Modeling of The First Trifurcation Made in Nepal", journal of hydro Nepal, issue no.5, july, 2009.

[2] C.A.AGGUIRE, R. G. RAMIREZ CAMACHO, "Head Losses Analysis in Symmetrical Trifurcation of Penstocks-High Pressure Pipeline Systems CFD", journal.

[3] Ivana BUNTIC, Thomas HELMRICH, Albert RUPRECHT"Very Large Eddy Simulation for Swirling Flows with Application in Hydraulic Machinery", journal.

[4] D. Bhandari, Dr. S. Singh, "Analysis of Fully Turbulent Flow in a Pipe using Computational Fluid Dynamics", International journal of Engineering research and Technology (IJERT),ISSN:2278-0181, Vol.1, Issue 5, July-2012.

[5] Basappa Meti, Prof. Nagaraj Sitaram, "Determination of Optimum Pressure loss and Flow Distribution at Pipe Trifurcation", journal of information, knowledge and research in civil engineering, ISSN 0975-6744.

[6] Mukesh Didwania, Lokesh Singh, Ashish Malik and Mangal S Sisodiya, "Analysis of turbulent flow over 90^0 bend of ducts using in centralized A.C plant by CFD code", IOSR journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 4 Ver. I (Jul- Aug. 2014), PP 41-48.

[7] Benny KUAN, William YANG, Chris SOLNORDAI and Phil SCHWARZ, "Dilute gas solid flow in mill-duct bifurcation: CFD simulation and experimental validation", Fifth International conference on CFD in the process Industries, CSIRO, Melborne, Australia, 13-15, December 2006.

[8] Sajid Hussein Ali Al – Abbasi, " CFD analysis of enhancement of Turbulent flow heat transfer in a Horizontal circular tube with Different inserts", European Scientific Journal May 2014 edition vol.10, No.15 ISSN: 1857 – 7881 (Print) e - ISSN 1857- 743.

[9] Y.S Chen and S.W Kim, "Computation of turbulent flows using an extended $k-\varepsilon$ turbulence closure model", October 1987.

[10] John D. Anderson, JR. "Computational fluid dynamics - The basics with applications", TATA McGRAW HILL edition.