iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 7 **Issue:** V **Month of publication:** May 2019

DOI: https://doi.org/10.22214/ijraset.2019.5450

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue V, May 2019- Available at www.ijraset.com

Characterization of the Fluid Flow Splitting through the Four-Way Connector for Industrial Application

Rahul Nag[1], Pratik Raj[2], Piyush Anand[3], Puneet Arora[4]
[1, 2, 3, 4]Department of Mechanical Engineering, University of Petroleum and Energy Studies, India

Abstract: This project focuses on the effect of splitter on fluid flow characteristics past a four-way-connector for different geometries and reports it to the scientific community. The aim of this work is to study flow properties at the junction of four-way-connector (FWC), minor loss and pressure loss suffered by the flow after passing through FWC. In this we have first designed and compared our results in FLUENT with classical formula and made an attempt to determine accurately the fluid flow characteristics. In this work we have studied head loss in junction of FWC with various inlet velocities, head loss in junction of FWC for different relevant non-dimensional geometrical parameters and FWC with different area of cross-section of the connector, and its impact on flow through different streams. In this work we will develop an experimental setup using 3-D printing and observe flow properties inside the junction and study the head loss suffered by fluid flow after passing through the junction. When the fluid flows past a junction there is some loss in the velocity and pressure head at the outlet. The losses depend on the various factors like angle between the connectors, physical properties of mixing liquid (viscosity), internal geometry of connector, diameter of pipes, etc. These minor losses can be minimized by properly designing the FWC junction of required geometry and for different diametric ratios of pipes and angle between them. The FWC has found many applications in the chemical industry where mixing of different substances of different phase (liquid with liquid, liquid with gas) takes place at different ratios. FWC will also be used for irrigation applications to distribute water properly in all the channels without much head loss. Similarly, the FWC can be used in water and sewage treatment plants to cater the heavy flow of streams.
Keywords: fluid flow, four-way-connector, CFD, chemical industry, minor loss, FLUENT.

I. INTRODUCTION

In any industry where transportation of fluid takes place by virtue of a pipeline system a numerous cross connection of pipes can be seen [1].Pipe networks are very common in industries, where fluid 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 the system. It is also sometimes essential to place valves, pumps or turbines of certain capacity to control pressure in the network. For these connections the losses at the junction can be significant and the process of handling these junctions can answer for the mass flow rate and pressure head calculations for the system. To outline the issue of concern this paper will focus on cross junctions which are widely seen in water transportation systems where there may be limited contact and retention time between the water flows into the incoming pipe legs. [1]

There have been enormous amount of study on finding the losses in various types of pipe connecting configurations since 1970s due to their enormous use in the municipal water distribution systems where they use cross, tee or wye- junction connectors [2]. Petroleum industry craves to raise concern about the research in this field. The determination of correct loss coefficient is vital for correctly designing the cross joint. The minor losses caused by the joints cause flow separation on the walls [3] following the secondary flows in the bend pipes.

In fluid dynamics, head is the difference in elevation between two points in a column of fluid, and the resulting pressure of the fluid at the lower point. It is possible to express head in either units of height (e.g. meters) or in units of pressure such as Pascals. When considering a flow, one says that head is lost if energy is dissipated, usually through turbulence; equations such as the Darcy-Weisbach equation have been used to calculate the head loss due to friction; the loss is proportional to square of the velocity. The laminar flow playing a major factor in determining unknown constant values for any cross connecting system which can lead to proper designing of the joint and correct loss coefficients. This is what Bernoulli demand in a steady flow, the sum of all forms of energy in a fluid along a streamline is the same at all points on that streamline. At the point when a fluid flows through a pipe, the

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue V, May 2019- Available at www.ijraset.com

fluid encounters a few resistances because of which some energy of the fluid is lost. [4] The conduct of pipe flow is legislated fundamentally by the impacts of viscosity and gravity with respect to the inertial forces of the flow. Contingent upon the impact of viscosity in respect to inertia, as spoke to by the Reynolds number, the flow might be either laminar or turbulent. This requires that the sum of kinetic energy, potential energy and internal energy remains constant. This takes us to the doorstep for determining loss coefficients for individual legs of the joint and correctly analysing the system. [6]

Since there are a number of possibilities for the flow in a cross joint the hydrodynamic analysis of the joint is difficult. There are four possibilities of flowing across: (1) Flow into one leg and out of three legs. (2) Flow into three legs and out of one leg. (3) Flow into two perpendicular legs and out of two perpendicular legs. (4) Flow into two opposite legs and out of two opposite legs. (Sharp et al. 2010). For any given flow condition, three K-factors are required to completely analyse across. Once the K-factors for each leg of a cross are acquired, this essential energy loss information can be used for analysis in hand calculations or in a computer software package. This study provides loss coefficients for each leg of a cross, resulting in an increased understanding of the head loss performance of crosses in pipe networks. [6]

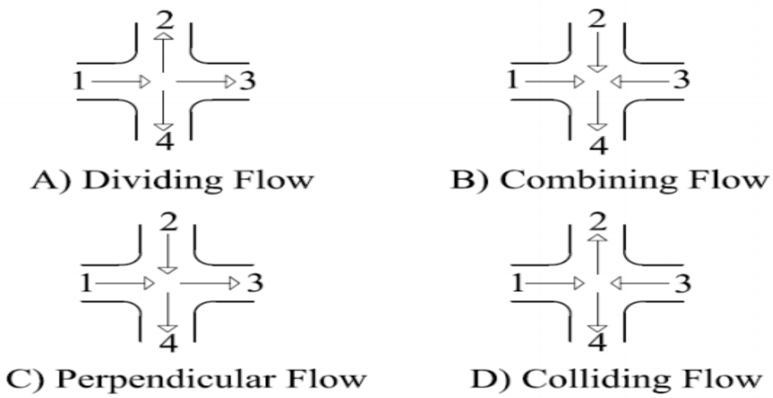


Figure 1 Different types of flow cases [8]

Losses in a piping system are typically categorized as major and minor losses. Minor losses in piping systems are generally characterized as any losses which are due to pipe inlets and outlets, fittings and bends, valves, expansions and contractions.

1. Major Energy Losses (This loss is due to friction)
2. Minor Energy losses [7]

Multiphase flow occurs when more than one material is present in a flow field and the materials are present in different physical states of matter or are present in the same physical state of matter but with distinct chemical properties. [8] The materials present in multiphase flow are often identified as belonging to the primary or secondary phases. The primary phase is defined as the phase that is continuous or enveloping the secondary phase. The secondary phase is the material that is distributed throughout the primary phase. The study of multiphase flow is very important in energy-related industries and in many industrial processes. Multiphase flows are also an important feature of our environment such as rain, snow, fog, mud slides, sediment transport, and debris flows etc. The simplest case of multiphase flow is two-phase flow. Two-phase flow can be solid-liquid flow, liquid-liquid flow, gas-solid flow, and gas-liquid flow. Pipelines are widely used in industrial applications for the transport of multiphase mixtures of liquid phases or gas and liquid phases. Here a small and common component of pipe network is considered for the study: FWC. FWC is a small, but very important part of the pipe-line system. It can vary in shape and it is primarily used for dividing or combining of flow in pipeline. The FWC can be also used for mixing of two different liquids, liquid and gas or two different gases. Junctions between pipes can involve the mixing or splitting of fluids. When the split at junctions involves more than one phase, the process becomes complicated since the ratio of phases at inlet and outlet are different.

Numerical studies related to two phase flow through FWC have been started long back. Experiments have been conducted for the investigation of the two-phase flow structure in the vicinity of the junction using the void probe technique developed by Herringe and Davis to investigate velocity, [9] void fraction and bubble size distributions within the flow. When a two phase flow enters a FWC, phase separation will often occurs. The lighter phase preferably gets diverted into the side arm and the heavier phase will flow towards the main arm. Hence the side arm of the dividing FWC will carry a higher proportion of the gas than the straight arm.

Fluid Structure Interaction (FSI) occurs when fluid flow causes deformation of the structure. [10] This deformation, in turn, changes the boundary conditions of the fluid flow. In FSI an important region is the interface surface which separates the fluid and the solid domain. At this interface surface, both the governing equations and the boundary conditions from the fluid and solid domain must be satisfied simultaneously. From previous literatures it is clear that three liquid-pipe interaction mechanisms can be distinguished: friction coupling, Poisson coupling and junction coupling. Friction coupling represents the mutual friction between liquid and pipe. Poisson coupling relates the pressures in the liquid to the axial (longitudinal) stresses in the pipe through the radial contraction or expansion of the pipe wall. Friction and Poisson coupling act along the entire pipe, junction coupling acts as specific points in a pipe system such as unrestrained valves, bends and tees.

Previous studies showed that that detailed studies were conducted on two phase flow through FWC. CFD studies were conducted on two phase flow. Fluid Structure Interaction studies were also conducted in piping systems. But studies incorporating CFD modelling of two phase flow through FWC with Fluid Structure Interaction is not reported. Hence this paper focuses on the study of redistribution of phases when multiphase flow enters a FWC and also to study the deformation occurs in pipe using Fluid Structure Interaction.

II. SIMULATION METHODOLOGY

FWCs are commonly encountered in pipeline systems. In this study a FWC with a horizontal branch is used with 5 inch length for each arm having inner diameter 1inch and 0.5inch. There has been no standard design available for four-way-connector (FWC). The losses in current available FWC are high. No scientific data is available related to minor losses in FWC.

When a multiphase flow enters a FWC a redistribution of phases often occurs. This redistribution can be desirable for certain situations where phase separation is required. For some situations this may lead to reduce the efficiency of pipeline systems. Hence it is necessary to study the phase separation phenomenon. The problem can be divided into two cases. Case 1: Simulation while considering change in velocity keeping physical properties of the fluid constant. Case 2: Simulation while considering constant velocity and varying radius of curvature of the joint.

The geometry of fluid domain of FWC was created on ANSYS FLUENT to study the phase separation phenomenon occurring on fluid when multiphase flow enters a FWC. The cross sectional area of one arm was named as inlet and the cross sectional areas of rest of the arms were named as outlet-1, outlet-2 and outlet-3 respectively in case of 1 inlet 3 outlet. Similarly, for the case of 3 inlet 1 outlet; 2 inlet 2 outlet orthogonally and anti-parallel flow. The geometry of fluid domain is shown in Fig. 2 to study the FSI the geometry of pipe was also created along with the fluid domain using ANSYS FLUENT 14.0 and Static Structural. The pipe has inner diameter 1 inch with 5 inch length for each arm. The pipe material used was structural steel. The geometry of pipe created is shown in Fig. Proper meshing was provided on the ANSYS Meshing stage as per the requirement of the problem.

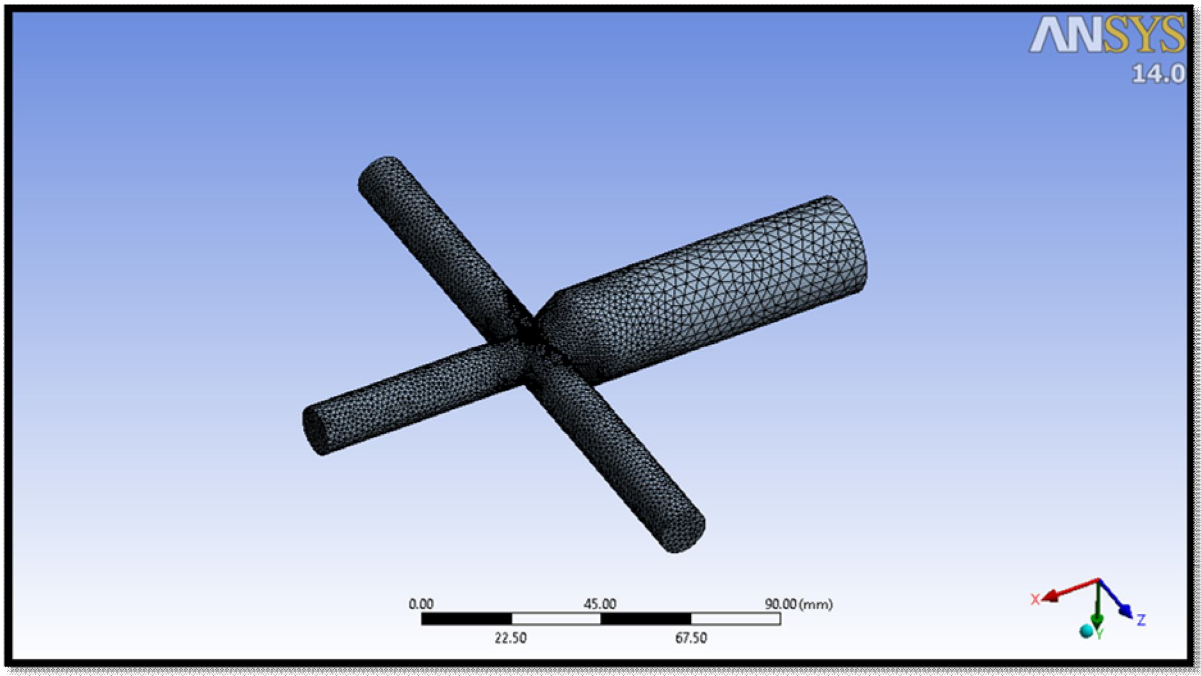


Figure 2 Mesh generation

For the phase materials the primary phase was taken as water. For the inlet the velocities of water must be given. The velocities of water at inlet were given as 0.05, 0.07, 0.1, 0.12, 0.15, 0.2, 0.5, 1, 5m/s respectively. At the outlet since the value of pressure was unknown, outflow boundary condition was given for all the outlets. Area weighted method is used to calculate the required values at both inlet and outlet.

Simulations were performed on the geometry created as mentioned above by applying the boundary conditions. Simulation for fluid domain was performed on ANSYS FLUENT. Realizable laminar and K-epsilon model were taken as the viscous model. This step involves feeding all prerequisite data such as pipe roughness, inlet velocity, type of flow, type of fluid, flow percentage, initializing the flow, number of iterations etc. Once the data are provided, calculation is done and immediately the iteration begins. When all the points converge, the calculation stops. After solution step, result step includes obtaining the results like pressure difference at inlet and outlet, net pressure in the pipe, outlet velocity etc. These results are used to calculate loss in the pipe and subsequently friction coefficient.

Grids play an equal and important role as the solver, in generating accurate results. The gridding methodology chosen (structured/unstructured/Cartesian, etc.) [11] for computation is not the only choice that matters, but also the spatial resolution of the flow field. The grids so generated may still be qualitatively fine and the solver will produce results with some variation in results among the grids, but it doesn't mean grid independent solution has been achieved. Unfortunately, this wrong understanding of GCS grids is strongly prevalent among many CFD practitioners, undermining the CFD results they generate.

Once the family of grids is generated, simulations are done on each grid and the flow field parameters of interest say, CL, CD, etc. are plotted against the grid size. Usually the change is value from tiny to coarse to medium are large. [12] Starting from the fine grid, the delta change in solution becomes small, asymptoting to a grid independent solution. From the plotted graph, one can pick the smallest of the grid, which gives grid independent solution for routine production runs. This is the optimal grid, which will provide the right solution with minimal solver run time.

GCS are done in CFD to make sure that the results obtained from simulations are due to the boundary conditions and the physics used and not because of mesh resolution. If the solution is invariant of the grid density, then we have achieved grid independence. [13]

Mesh Independence Study is conducted to ensure that the solution obtained is independent of the mesh resolution. [14] This study is carried out for round edge model with various mesh sizes. The results show that there is only 0.5% of differences in loss coefficient with mesh size increment after 4.48 million elements mesh size. Thus 4.48 million elements mesh size was chosen in the CFD analysis and similar mesh scheme was adopted for square edge model.

The flow of most fluids can be mathematically described by the use of continuity equation and momentum equation. According to continuity equation, the amount of fluid entering in certain volume leaves that volume or remains there and according to momentum equation tells about the balance of the momentum. The momentum equations are sometimes also referred as Navier-Stokes (NS) equation. They are most commonly used mathematical equations to describe flow. [15] In this section we shall first derive NS equations and then K-Epsilon model. For incompressible fluids the Continuity Equation is

$$\partial u/\partial x + \partial v/\partial y + \partial w/\partial z = 0$$

Navier-Stokes (NS) equation derived by considering three directions of flow x, y and z

$$\{\rho\,\partial u/\partial t + \rho u\,\partial u/\partial x + \rho v\,\partial u/\partial y + \rho w\,\partial u/\partial z\}\Delta x\Delta y\Delta z = \Sigma\, Fx$$

K-epsilon model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho \in - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho \in) + \frac{\partial}{\partial t}(\rho \in u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \in}{\partial x_j}\right] + C_{1\epsilon}\frac{\in}{k}(P_k + C_{3\epsilon}P_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

SIMPLE SCHEME:

$$p^{k+1} = p^k + urfp$$

$$m_f^{k+1} = m_f^* + m_f'$$

$$\vartheta^{k+1} = \vartheta^* - \frac{Vol\nabla p'}{a_p^v}$$

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue V, May 2019- Available at www.ijraset.com

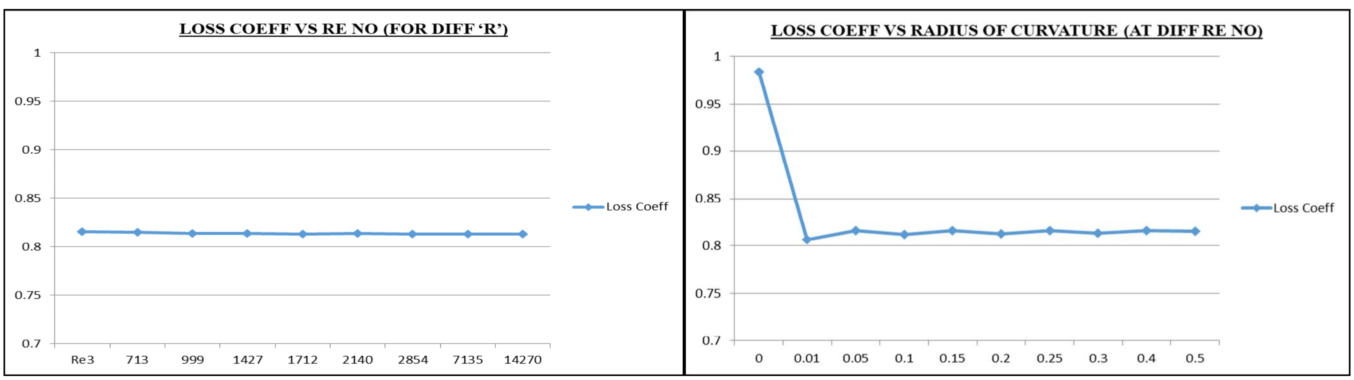
III. RESULTS AND DISCUSSION

The computational experimental setup was designed and performed in FLUENT. CFD Analysis was performed and sufficient data was generated and analysed, to establish Velocity and pressure characteristics for the water flow through cross junction (FWC) of pipe, and to determine minor losses. The main conclusions of the present work are as under.

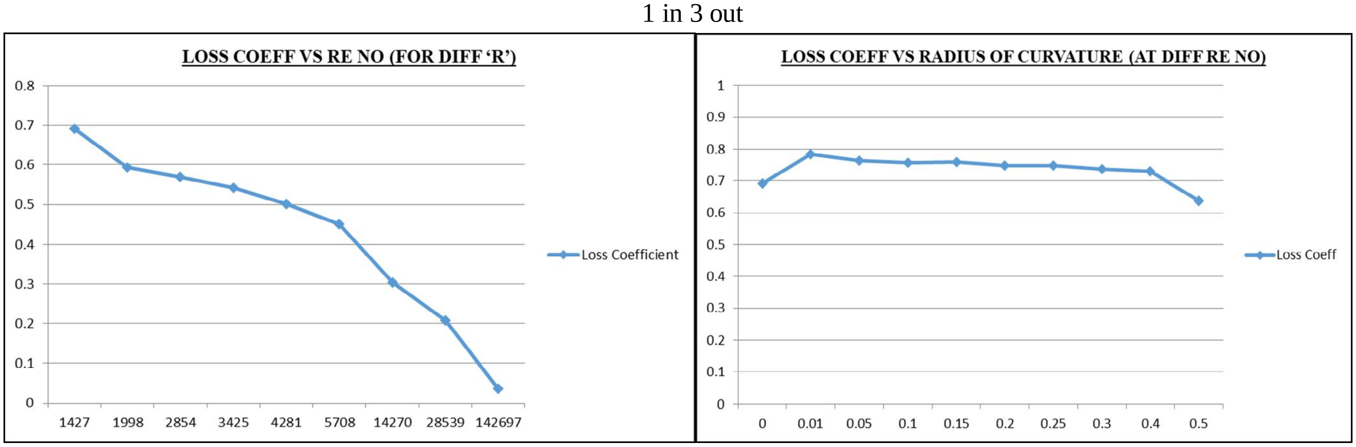
A. In case of 3in 1out, minor loss coefficients remains constant with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficient first decreases exponentially and remains constant.
B. In case of 1in 3out, minor loss coefficients decreases with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficients remains almost constant with about 5% fluctuations in maximum and minimum value.
C. In case of 2in 2out orthogonal flow, value of minor loss coefficients remains constant with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficient first decreases gradually.
D. In case of 2in 2out anti-parallel flow, value of minor loss coefficients remains constant with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficient first decreases gradually.

The approach of the computational solution to the self-similar exact solution in this test case depends more on grid resolution than on locations and distances. As described, the coarser the grid, the more smeared the expansion fan, especially near the corner. However, the overall property ratios across the FWC should still be accurately satisfied for reasonably fine grids. The degree of spatial uniformity of the solution on either side of the FWC should also be high away from the corner region and the downstream flow originating in that region, even on coarse grids. For higher-order computational schemes, any oscillations in the solution should be relatively slight, especially around the edges of the FWC, and the amplitude of any such oscillations should not continue to increase with increasing grid resolution. The amplitudes of any oscillations around the edges of the FWC should not be more than 5% of the total change in the corresponding total property change across the FWC, especially well away from the corner.

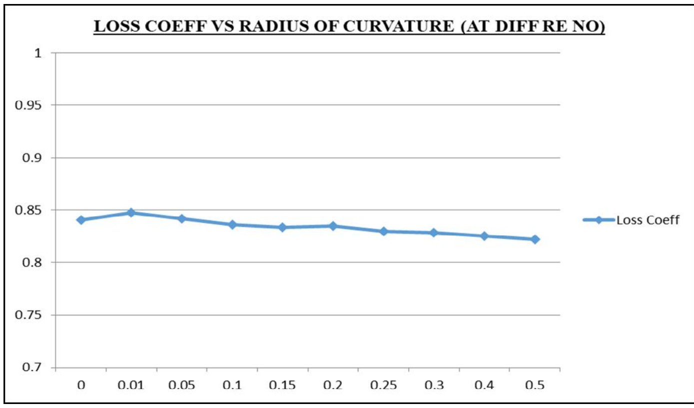
3 in 1 out

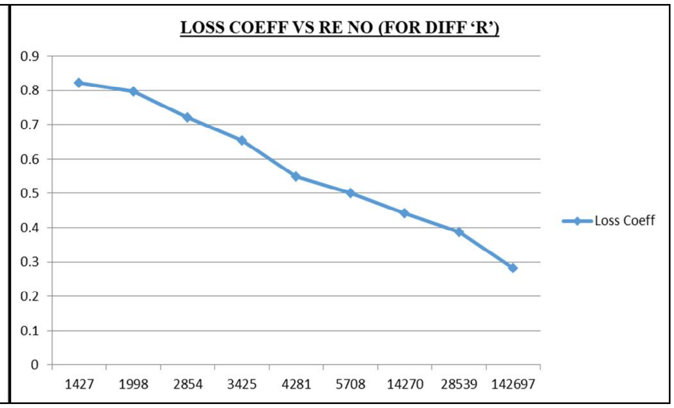


1 in 3 out

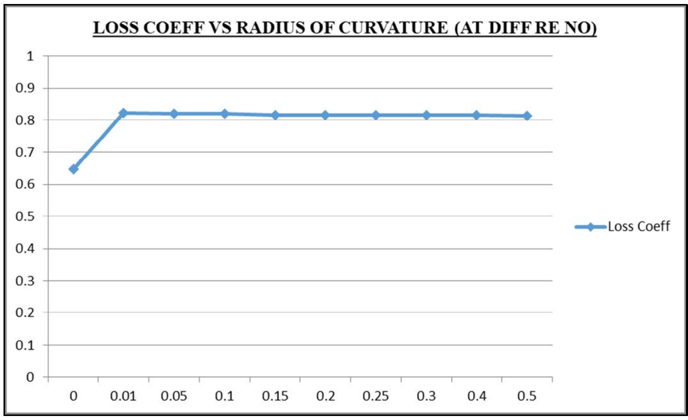


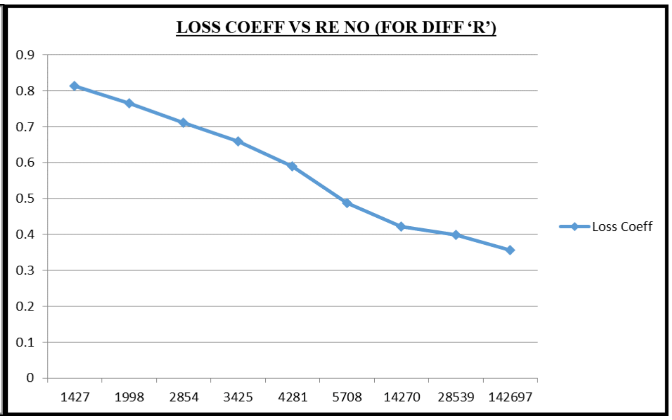
2 in 2 out orthogonal





2 in 2 out Anti parallel





In this case the validation of CFD simulation is done by using an elbow joint. The elbow joint is designed in ANSYS Fluent and simulations are done for all the same data points and under same conditions (meshing and fluent model). The results obtained from the analysis are compared with the experimental values of loss coefficient for the same elbow joint. The error comes out to be 2.8%

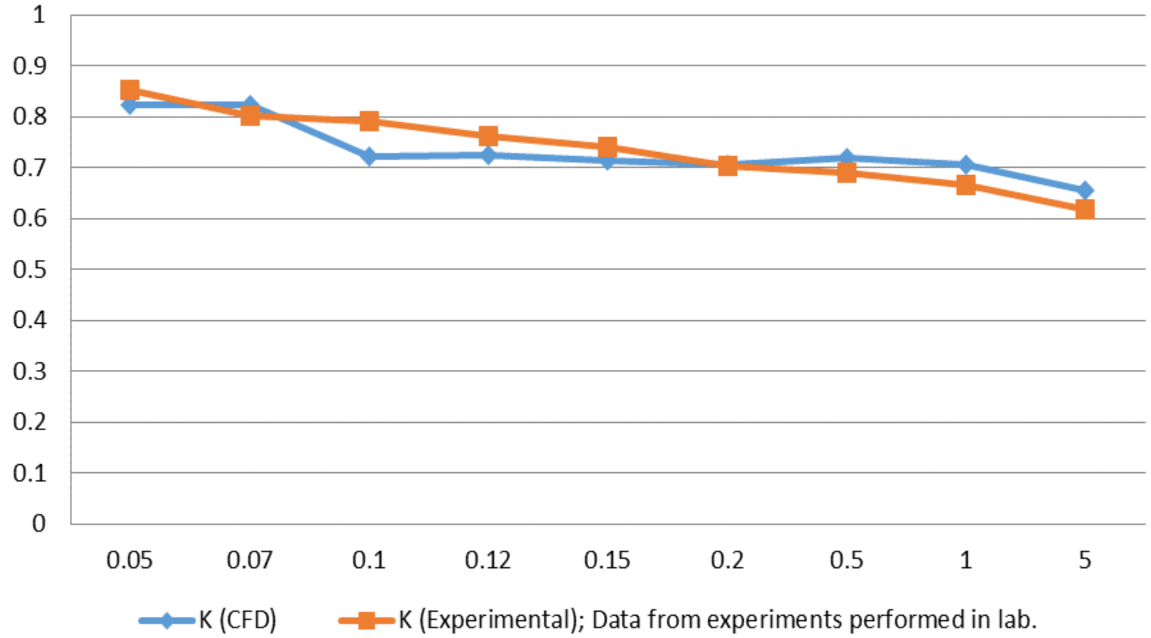


Figure 3 Plot between experimental and computational data

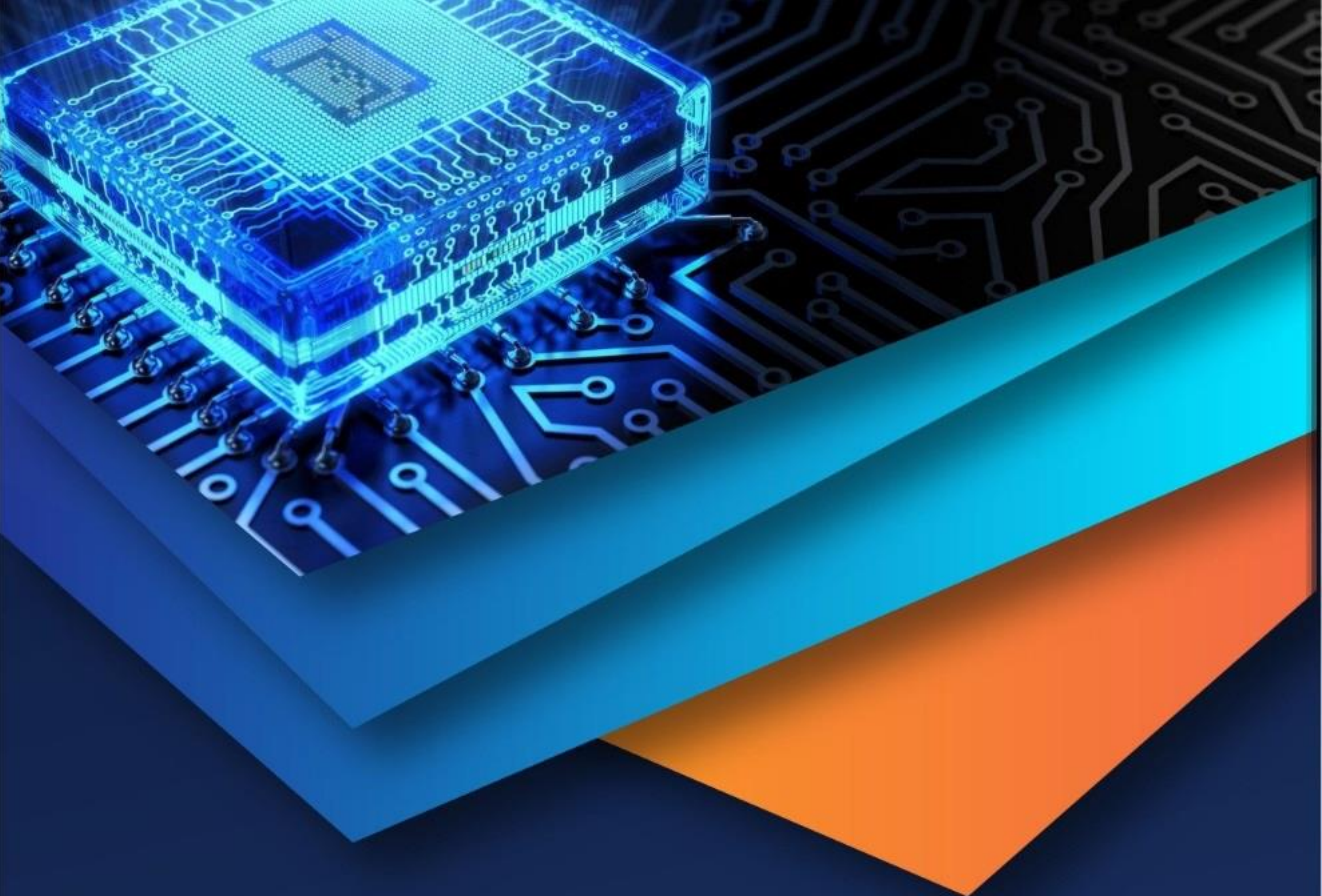
IV. CONCLUSIONS

The computational experimental setup was designed and performed in FLUENT. CFD Analysis was performed and sufficient data was generated and analyzed, to establish Velocity and pressure characteristics for the water flow through cross junction (FWC) of pipe, and to determine minor losses. The main conclusions of the present work are as under.

A. In case of 3in 1out, minor loss coefficients remains constant with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficient first decreases exponentially and remains constant.
B. In case of 1in 3out, minor loss coefficients decreases with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficients remains almost constant with about 5% fluctuations in maximum and minimum value.
C. In case of 2in 2out orthogonal flow, value of minor loss coefficients remains constant with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficient first decreases gradually.
D. In case of 2in 2out anti-parallel flow, value of minor loss coefficients remains constant with increasing Reynolds no at a particular D/d ratio. And with radius of curvature (D/d) ratio, minor loss coefficient first decreases gradually.

REFERENCES

[1] D. P., and Oberkampf, W. L. Aeschliman, "Experimental Methodology for Computational Fluid Dynamics Code Validation," AIAA Journal, vol. 36, no. 5, pp. 733-741, 1998.
[2] D. P., Oberkampf, W. L., and Blottner, F. G. Aeschliman, "A Proposed Methodology forCFD Code Verification, Calibration, and Validation," in 16th International Congress on Instrumentation for Aerospace Simulation Facilities, Dayton, OH, 1995, pp. Paper 95-CH3482-7.
[3] Anand B. Desamala, "CFD Simulation and Validation of Flow Pattern Transition Boundaries during Moderately Viscous Oil-Water Two-Phase Flow through Horizontal Pipeline," World Academy of Science, Engineering and Technology, no. 73, 2013.
[4] C. Udaya Kiran Aslam A. Hirani, "CFD Simulation and Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe," IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), vol. 10, no. 1 (Nov. - Dec. 2013), pp. 31-34.
[5] Boris Hu er, "CFD simulation of a T-junction," Institute of Hydraulic and Water Resources Engineering, Department of Hydraulic Engineering, Vienna University of technology, Austria
[6] D. Bhandari and Dr. S. Singh, "Analysis Of Fully Developed Turbulent Flow In A Pipe Using Computational Fluid Dynamics”," International Journal of Engineering Research & Technology (IJERT), vol. 1, no. 5, July 2012.
[7] Fernando T Pinho and Rodrigo Maia Gyorgy Paal, "Numerical Predictions Of Turbulent Flow In A 90° Tee Junction," in The 12th International Conference on Fluid Flow Technologies, Budapest, Hungary, September 3 - 6, 2003.
[8] Wahid S. Mohammed Jafar M. Hassan, "CFD Simulation for Manifold with Tapered longitudinal Section," International Journal of Emerging Technology and Advanced Engineering, vol. 4, no. 2, February 2014.
[9] Hiroshi Ogawa, "Experimental Study on Fluid Mixing Phenomena In T-Pipe Junction with Upstream El ow," in The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11) Popes Palace Conference Center, Avignon, France, 2005.
[10] Roman Klas and Oldich sperka Jaroslav stigler, "Characteristics of the T-junction with the equal diameters of all branches for the variable angle of the adjacent ranch”," in EPJ Web of Conferences, 2014.
[11] Dr. R. K. Bansal, "Fluid Mechanics and Hydraulic Machines," no. 6, 1997.
[12] Prof Harpreet Singh Prof. Balvinder Singh, "CFD analysis of Fluid Flow Parameters within A Y-Shaped Branched Pipe," International Journal of Latest Trends in Engineering and Technology (IJLTET).
[13] H. K. Versteeg and W. Malasekera, "An Introduction to Computional Fluid Dynamics," The finite volume method, Longman group Ltd., 1995
[14] Malandrone and Panella B Bertani C, "Two-Phase Flow in horizontal T-junction: pressure drop and phase separation," in International Conference on Heat Transfer, 2007, pp. 1-6.
[15] Christopher E. Brennen, "Fundamentals of Multiphase Flows," California Institute of Technology, 2005.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY