

# Study of Friction Factor of Pipe for Different Bend Angles and Bend Radius by Experiment and Simulation

Amal A Thomas<sup>1</sup>, Bipin Biju<sup>2</sup>, Jithin Raj Johnson<sup>3</sup>, Tito George Thomas<sup>4</sup>, Roshbe S Calolsa<sup>5</sup>

U.G. Student, Dept. of Mechanical Engineering, St Thomas College of Engineering and Technology, APJ Abdul Kalam Technological University, Kerala , India<sup>1</sup>

U.G. Student, Dept. of Mechanical Engineering, St Thomas College of Engineering and Technology, APJ Abdul Kalam Technological University, Kerala , India<sup>2</sup>

U.G. Student, Dept. of Mechanical Engineering, St Thomas College of Engineering and Technology, APJ Abdul Kalam Technological University, Kerala , India<sup>3</sup>

U.G. Student, Dept. of Mechanical Engineering, St Thomas College of Engineering and Technology, APJ Abdul Kalam Technological University, Kerala , India<sup>4</sup>

Assistant Professor, Dept. of Mechanical Engineering, St Thomas College of Engineering and Technology, APJ Abdul Kalam Technological University, Kerala , India<sup>5</sup>

**ABSTRACT:** Determination of friction factor in pipe flow is necessary for various applications dealing with fluid flow. The aim of this work is to investigate the friction factor along bend for locally available 1-inch PVC pipe. Friction factor for various bend angles and curvature ratios of these 1-inch PVC pipe bends are ascertained using Numerical analysis. The experimental method is also conducted for the validation of these results. Head losses are measured under a flow velocity of 2 m/s. The simulation of water flow in a bent pipe of 1-inch diameter and different bend radius and angle is done by solving the Navier-Stokes Equation, with the help of commercial simulation software ANSYS 16 (WORKBENCH and FLUENT). The  $k - \epsilon$  turbulence model is implemented in the computational fluid dynamics for the turbulent flow condition. As Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) algorithm is used Navier-Stokes equations for viscous, incompressible flow shows mixed elliptic- parabolic behavior. Variation of wall shear is calculated at the inner and outer bends which are used to calculate the variations of friction factor along the bends. The study is conducted in curvature ratios 2 and 5 and for bend angles 60°, 90°, and 180°. The friction factor obtained is presented graphically in contrast to inner and outer curve lengths. Comparison of graphs is also performed to understand the variation in friction factor for various bend angle and curvature ratios. The work involved culminating in an efficient pipe design useful in any fluid flow application.

**KEYWORDS:** Computational Fluid Dynamics, Dissipation rate, Turbulent viscosity, Centripetal Acceleration

## I. INTRODUCTION

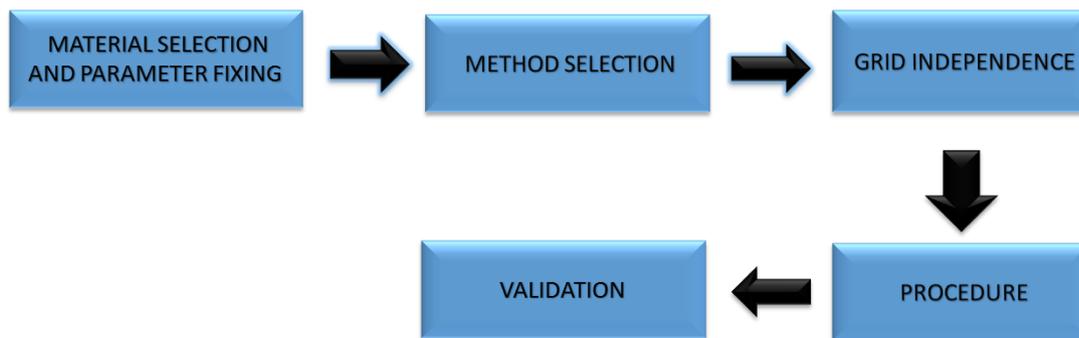
A pipe is a hollow cylinder or tubular section, usually but not necessarily of the circular cross-section using mainly to convey substances which can flow- liquids and gases, slurries, powders, etc. The pipe is generally specified by a nominal diameter with constant outside diameter and a schedule that defines the thickness. Pipe fittings like elbows, bends, and valves are essential in many industrial applications such as hydraulic, nuclear and chemical engineering systems and the prediction of the pressure drop across these fittings is crucial in the design of piping systems. Valves control fluid flow and regulate pressure. Fluid flowing through pipe experiences some resistance due to which some of the energy of the fluid is lost. This loss of energy is classified as the losses due to friction and the losses due to sudden expansion, contraction, fittings, etc. The losses due to friction are called Major loss or frictional losses and losses due to fittings, sudden expansions, etc., are called Minor loss. Bends are provided in pipes to change the direction of flow through it. An additional loss of head, apart from that due to fluid friction, takes place in the course of flow through a pipe bend. The fluid takes a curved path while flowing through a pipe bend. Computational Fluid Dynamics (CFD) is

predicting what will happen, quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer, phase change, chemical reaction, mechanical movement, stresses in and displacement of immersed or surrounding solids. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid and then apply a suitable algorithm to solve the equations of motion.

## II. RELATED WORK

In the context of our study, there are some contributions of particular importance which led us to select and choosing parameters for our study. Scott Richards et.al [1] provided a tutorial containing instructions for meshing two internal flows and analyzing both of them in FLUENT. The first flow was in a straight pipe and the second flow was in a pipe of the same length and diameter but with 90° elbow. Jongtae Kim et.al [2] conducted a study on turbulent flow in a 90° elbow. In their study, they performed experimental and numerical work on turbulent flow in a pipe with a 90° elbow. In another study, PrasunDutta and Nityananda Nandi [3] conducted a study on the turbulent flow of single-phase incompressible fluid through 90° pipe bend, which had been simulated numerically using k-ε turbulent model. Anup Kumar Datta et.al [4] carried out a study on turbulent flow through helical pipes with circular cross-section was numerically investigated. SaswatSambit and Awadesh Kumar [5] conducted a study on the flow through a straight pipe. Simulation and flow analysis were conducted and the study shows that the more viscous the fluid more was the frictional coefficient and thus higher the frictional loss. A. Noorani et.al [6] studied the evolution of turbulence characteristics from straight to curved pipes. Direct numerical simulations had been performed in order to study the effect of curvature and Reynolds number in fully developed turbulent pipe flows

## III. METHODOLOGY



## IV. EXPERIMENTAL SETUP

The apparatus consists of a PVC pipe of 27.02 mm in diameter. A gate valve is provided at the outlet of pump which enables experiments to be conducted at different flow rates, i.e. at different velocities. Pressure tapings are provided along the length of pipes so that the drop of the head can be visualized along the length of the pipe. Fig.6 illustrate its CAD drawing respectively.

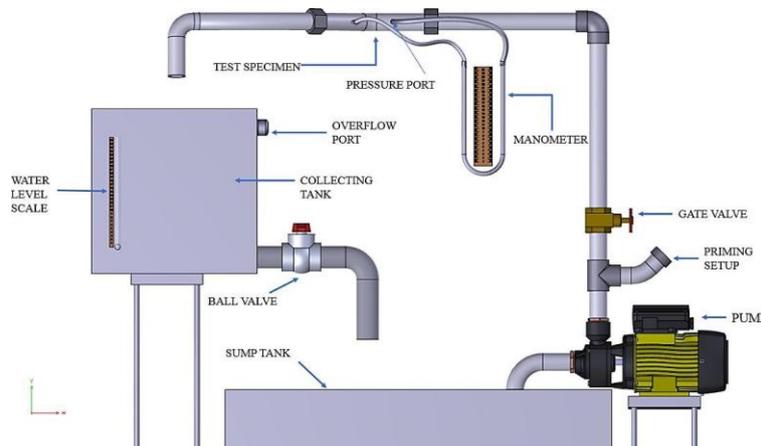


Fig.1 CAD drawing of Experimental setup

The components required for experimental setup are Water pump, Gate valve, Ball valve, Manometer ,Test specimen , Collecting tank and other components like foot valve, FTA, MTA, flexible hose. Connections were done as shown in Figure 1 .Sump tank was filled with water and priming was done. All the outlet valves were opened and the pump started. By closing all the outlet valves, for each pipe leakages were checked, and the correction was made for the leaks if any. All the outlet valves of the pipe to be tested were opened. All the air bubbles from manometer and connecting pipes were removed. The flow was reduced by adjusting gate valve, so that water heads in manometer are to the readable height. The heads and flow rate were observed. Now, the flow was increased by adjusting gate valve, so that water will not overflow. Switch off the pump after taking all readings.

**V. EXPERIMENTAL OBSERVATION**

MANOMETER READING			HEAD LOSS	TIME	VELOCITY	REYNOLDS NUMBER	PRESSURE
h <sub>1</sub>	h <sub>2</sub>	h <sub>3</sub>	H	t	v	R <sub>e</sub>	P
cm	cm	cm	m	s	m/s		Pa
15.30	15.75	0.45	0.0567	8.58	1.626	49364.62	556.227
15.25	15.84	0.59	0.0753	15.00	1.860	56468.76	739.411
15.10	15.89	0.79	0.1005	12.37	2.256	68491.14	985.881
15.15	16.04	0.89	0.1121	5.85	2.385	72407.52	1100.325
15.10	16.01	0.91	0.1145	5.70	2.447	74289.82	1123.458
15.00	15.99	0.99	0.1256	10.59	2.634	79967.05	1232.351

Table 1. Experimental observations

**Sample Calculation**

Density of Mercury,  $\rho_{Hg} = 13600 \text{ kg/m}^3$

Density of water,  $\rho = 1000 \text{ kg/m}^3$

Manometer Reading,  $h_1 = 15.10 \text{ cm}$ ,  $h_2 = 15.89 \text{ cm}$

Head difference,  $h = h_2 - h_1 = (15.89 - 15.10) = 0.79 \text{ cm of Hg} = 0.0079 \text{ m of Hg}$

Head Loss,  $H = \frac{(\rho_{Hg} - \rho)h}{\rho} = \frac{(13600 - 1000) \times 0.0079}{1000} = 0.1005 \text{ m of water}$

Pressure difference,  $P = H\rho g = 0.1005 \times 1000 \times 9.81 = 985.881 \text{ Pa}$

Time for 10cm rise of water in collecting tank,  $t = 12.37 \text{ s}$

Area of collecting tank,  $a = 0.16 \text{ m}^2$   
 Height of water rise,  $y = 0.1 \text{ m}$   
 Flow rate,  $Q = ay = 0.16 \times 0.1 = 0.0012934 \text{ m}^3/\text{s}$   
 $t = 12.37$   
 Area of pipe,  $A = \frac{\pi d^2}{4} = \frac{\pi \times 0.027022^2}{4} = 0.0005734 \text{ m}^2$   
 Velocity of flow,  $v = \frac{Q}{A} = \frac{0.0012934}{0.0005734} = 2.256 \text{ m/s}$   
 Reynolds Number,  $R_e = \frac{\rho v d}{\mu} = \frac{1000 \times 2.256 \times 0.02702}{0.000890} = 62491.14$

### VI. RESULT AND DISCUSSIONS

The results are organized into two sections for the ease of understanding.

- First section contains the graphs showing variation of friction factor on inner and outer bend v/s curve length and velocity contour for each separate bend angle (60°, 90°, 180°) for each curvature ratios (R/d= 2, 5).
- Second section deals with the variation of friction factor on inner and outer bend v/s curve length for each bend angle for different curvature ratio.

For simplicity 60° bend with curvature ratio (R/d) =2 is written as 60RD2 and for the other cases also. Bends are provided in pipes to change the direction of flow through it. An additional loss of head, apart from that due to fluid friction, takes place in the course of flow through a pipe bend. The fluid takes a curved path while flowing through a pipe bend. Whenever a fluid flows in a curved path there must be a force acting radially inwards on the fluid to provide the inward acceleration, known as centripetal acceleration. This results in an increase in pressure near the outer wall of the bend.

#### • First Section

##### 60RD2

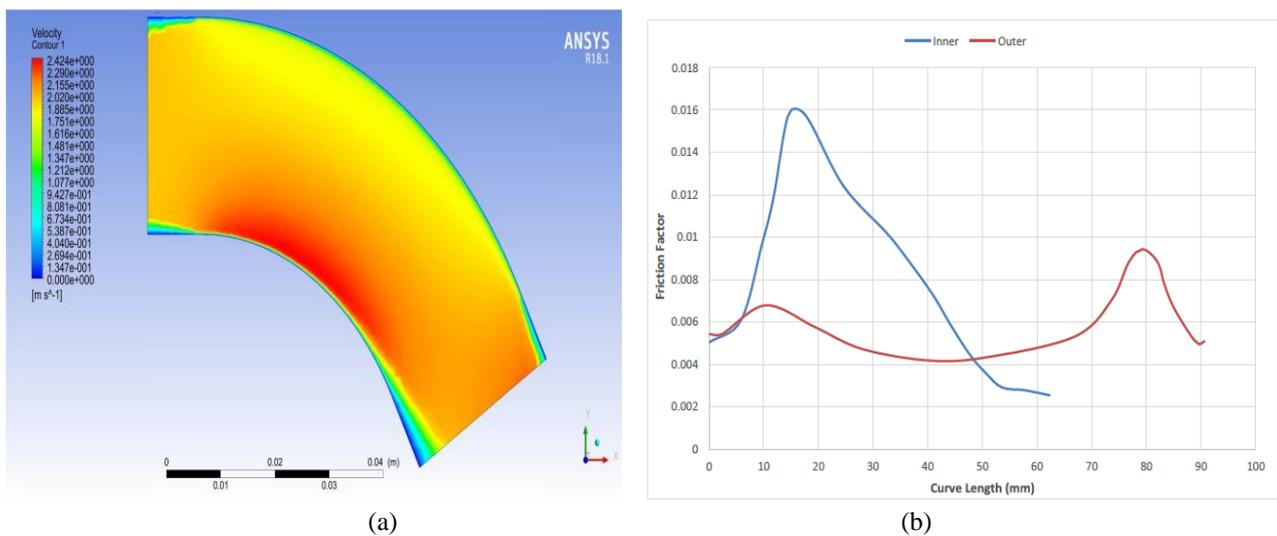


Fig.2.60RD2(a)Velocity Contour for 60RD2 (b) Friction Factor v/s Curve Length for Inner and Outer Bend of 60RD2.

**90RD5**

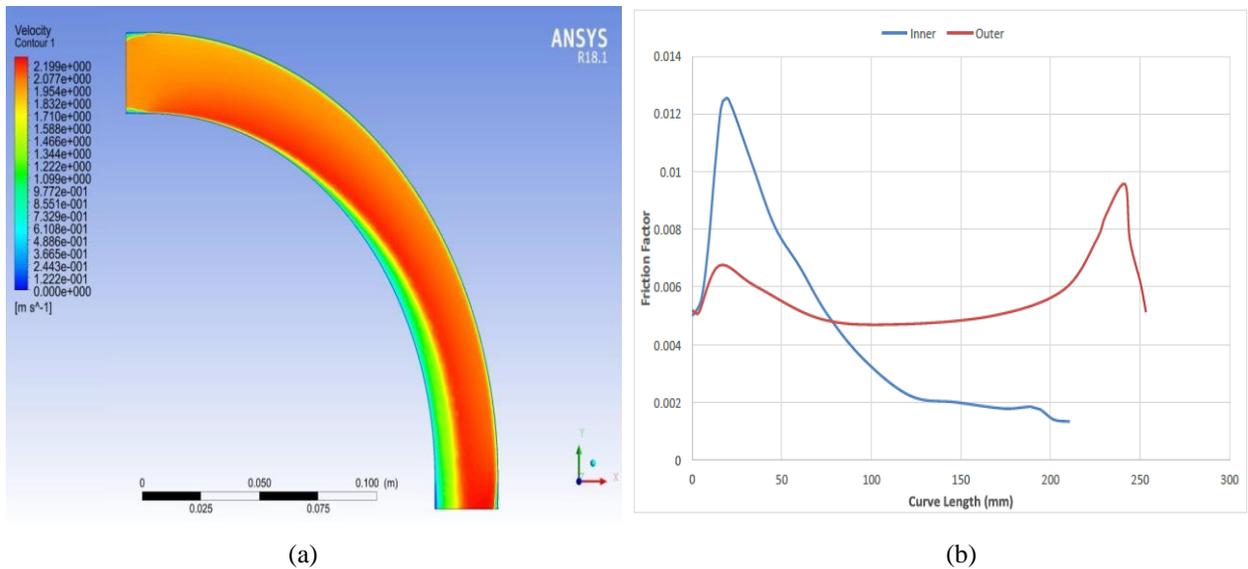


Fig.3. **90RD5** (a)Velocity Contour for 90RD5 (b)Friction Factor v/s Curve Length for Inner and Outer Bend of 90RD5.

**180RD2**

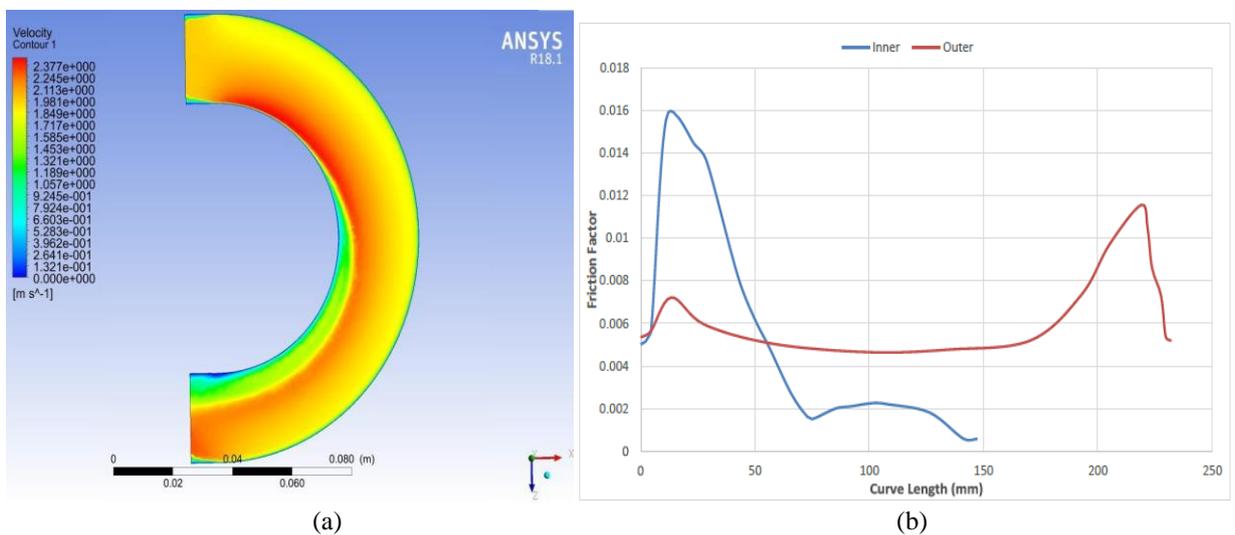


Fig.4. **180RD2** (a)Velocity Contour for 180RD2 (b)Friction Factor v/s Curve Length for Inner and Outer Bend of 180RD2

• Section 2

**90 Degree**

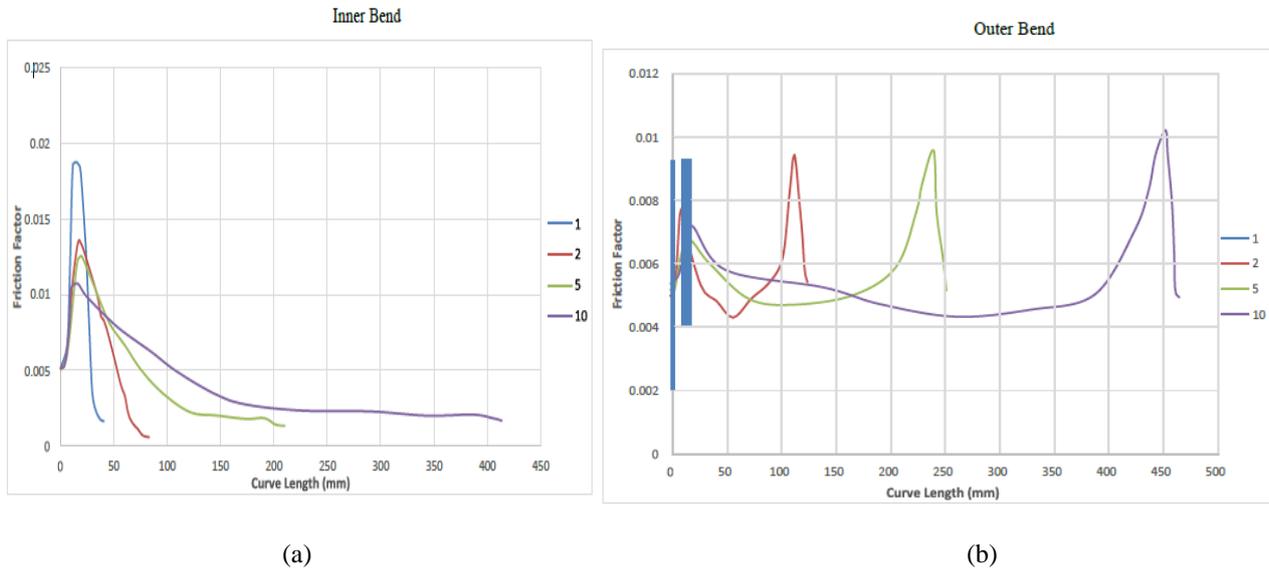


Fig.5.90 DEGREE (a) Friction Factor v/s Curve Length for Inner bend of 90° for different R/d. (b)Friction Factor v/s Curve Length for Outer bend of 90° for different R/d.

The Interpretation of the results showed that:

- As the bend angle increases Friction factor increases.
- Increase in curvature ratio decreases the Friction factor.

The velocity contour of a bend shows that at the inner bend the velocity is higher when compared to the outer bend. This is because whenever a fluid flows in a curved path, there must be a force acting radially inwards on the fluid to provide the inward acceleration, known as centripetal acceleration. This results in an increase in pressure near the outer wall of the bend, resulting in low velocity at the outer bend.

**VII .CONCLUSION**

From the study of friction factor in locally available 1-inch PVC pipe of bend angles 60°, 90°, and 180° and curvature ratios 2 and 5 it can be concluded that:

- As the bend angle increases Friction factor increases.
- Increase in curvature ratio decreases the Friction factor.
- Velocity is higher in the inner bend when compared with the outer bend. This is because whenever a fluid flows in a curved path (Fig.7.1), there must be a force acting radially inwards on the fluid to provide the inward acceleration, known as centripetal acceleration. This results in an increase in pressure near the outer wall of the bend, resulting in low velocity at the outer bend.
- Towards the exit section after the bend, a secondary flow is developed which causes a decrease in friction factor at the inner bend and in-order to keep the total energy constant friction factor increases at the outer bend.

Thus, the following conclusions can be drawn from the graph that the optimum bend can be designed for various application . Usage of bends should be avoided where ever possible, as it adds up extra pressure head. From the results obtained from numerical analysis, bend selection depends on angle and curvature ratio. If we have to choose bends between different angles, select the minimum possible degree of an angle as it shows a minimum variation of friction factor. If we have to choose bends based on curvature ratio; if more space is available to consider selecting bends with large curvature ratio. Thus, the friction factor can be minimized by selecting the minimum possible degree of angle and large curvature ratio.

#### REFERENCES

- [1] Scott Richards, Keith Martin and John M Cimbala (2010), ANSYS Workbench tutorial- Minor losses, Penn State University, 1-11.
- [2] Jongtae Kim, Mohan Yadav and Seungjin Kim (2014), Characteristics of Secondary Flow Induced by 90° Elbow in Turbulent Pipe Flow, Engineering Applications of Computational Fluid Mechanics, 8:2, 229-239.
- [3] PrasunDutta and Nityananda Nandi (2015), Effect of Reynolds Number and Curvature Ratio on Single Phase Turbulent Flow in Pipe Bends, Mechanics and Mechanical Engineering, Vol. 19, No. 1, 5–16.
- [4] Anup Kumar Datta, Yasutaka Hayamizu, ToshinoriKouchi, Yasunori Nagata, Kyoji Yamamoto and Shinichiro Yanase (2017), Numerical study of turbulent helical pipe flow with comparison to the experimental results, Journals of Fluids Engineering, ASME.
- [5] WenshanPeng and Xuewen Cao (2016), Numerical prediction of solid particle erosion in pipe bends with the liquid-solid flow, Powder Technology.
- [6] J. P. Powers, A. B. Corwin, Paul C. Schmall and W. E. Kaeck (2007), Construction Dewatering and Groundwater Control: New Methods and Application, Wiley and Sons, Inc. ISBN: 978-0-471-47943-7, 597-602.
- [7] PrasunDutta, Sumit Kumar Saha, Nityananda Nandi And Nairit Pal (2016), Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modeling, Engineering Science and Technology, an International Journal, 19, 904-910.
- [8] K. Sudo, M. Sumida and H. Hibara (1998), Experimental investigation on turbulent flow in a circular-sectioned 90° bend, Experiments in Fluids, 42-49.56
- [9] Manish Kumar Rathore, Dilbag Singh Mondloe and SatishUpadhyay (2017), Computational Investigation of Fluid Flow 90° Bend Pipe Using Finite Volume Approach, International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 06, 56-72.
- [10] N Crawford, S Spence, A Simpson And G Cunningham (2009), A numerical investigation of the flow structures and losses for turbulent flow in 90° elbow bends, Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering, Vol.223, 27-44.
- [11] Aslam A. Hirani and C. UdayaKiran (2013), IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Volume 10, Issue 1, 31-34.
- [12] EbrahimAhmadloo, NajmehSobhanifar and Fatemeh Sadat Hosseini (2014), Computational Fluid Dynamics Study on Water Flow in a Hollow Helical Pipe, Open Journal of Fluid Dynamics, 133-139.