# CFD Analysis of Fluid Flow through Elbow to Optimize the Design of Elbow

**Dr. Satypal T. Warghat[1] Pavan J. Bhadange[2] Manoj. J. Watane[3]**
[1]Head of the Dept. [2,3]Assistant Professor
[1,2,3]Department of Mechanical Engineering
[1,2,3]Dr. Sau. Kamaltai Gawai Institute of Engineering and Technology, Darapur, Daryapur, Amravati, Maharashtra 444814, India

*Abstract*— Elbow are the pipe fittings of engineering as well as domestic application used in wide range of size. This report deals with the computational fluid dynamics analysis of flow through elbow. This involves with 2D analysis of flow through pipe fittings. CFD analysis are performed in two different diameters elbow. The pressure drops at two different upstream & downstream locations are investigated using computational methods. The four different velocities like 0.75, 1.0,1.5 and 3m/sec and pipes of diameter 1", 1/2" are used in this study. CFD analysis is performed using K-ε model & FLUENT software is used for analysis purpose. And for pipe modeling GAMBIT 2.01 software is used. This study will help to predict flow losses through pipe fittings and help to design pipe lines. And also this study will helps to optimize the design of elbow.

***Keywords:*** Elbow, CFD, GAMBIT, FLUENT, K-ε Model

## I. Introduction

Elbow perhaps the most frequently used pipe fittings. Therefore pressure loss through such pipe fittings are of considerable engineering importance. However many engineers investigate the losses through pipe fittings but they reported widely varying result. This is probably due to their use of different inlet velocity distributions and also to the conditions at the outlet of the pipe.. The two pipes of diameters 1", 1/2" are used to determine the losses through elbow. CFD analysis is performed using K-ε model & commercial code FLUENT software is used for analysis purpose. And for pipe modeling GAMBIT 2.01 software is used. All the tests are made at approximately the same Reynolds number with approximately the same inlet velocity profiles, and in each case the downstream transition length is sufficient for a nearly symmetrical velocity distribution to be attained at the pipe outlet.

This study will help to predict flow losses through pipe fittings and help to design pipe lines and also useful to optimize the design of elbow.

## II. Literature review

Timothy A. Akintola & Solomon O. Giwa (2009) has developed A computer aided technique for determination of optimum pipe diameter for non viscous flow using the least cost approach. Results obtained from the validation of the developed software revealed that optimum pipe diameter for turbulent flow increases linearly with fluid compressibility [13].

Bohuslav Kilkovsky & Zdenek Jegla (2011) has compared different methods for calculating value of resistance coefficient K. He found that L/D method is overstating fitting pressure loss where as crane method has better agreement with 2K and 3K methods if we take them as good as benchmark. At low Reynolds number L/D method generates high resistance coefficient because of sudden increase in friction factor when flow regime goes towards laminar flow [14].

T. K. Bandyopadhyay & S. K. Das (2011) perform CFD analysis of Non Newtonian fluid through elbow and they found that the maximum velocity is shifted towards the inner wall of the elbow. The pressure is greatest at the outer wall furthest from the centre of curvature and least at the inner wall nearest to the centre of curvature. This is due to centrifugal forces [15].

Quamrul H. Mazumder (2012) was performed Computational fluid dynamics (CFD) analysis of single-phase and two-phase flow in a 90 degree horizontal to vertical elbow with 12.7 mm inside diameter. Characteristic flow behavior was investigated at six different upstream and downstream locations of the elbow. To evaluate the effects of different phases, three different air velocities and three different water velocities were used during this study. Pressure and velocity profiles at six locations showed an increase in pressure at the elbow geometry with decreasing pressure as fluid leaves from the elbow [16].

## III. Computational fluid dynamics

### A. Introduction to Computational Fluid Dynamics(CFD)

The tool used is computational fluid dynamics (CFD) to predict the flow rate, pressure and velocity at each point in pipe. The k-ε model is used is the most used model in the industries and it's enough strong and accurate for our calculations. This tool enables us to do several simulations rapidly. In this analysis the single phase models model is used for solving the respective category problems. This model will calculate one transport equation for the momentum and one for continuity for single phase, and then energy equations are solved to analyze the behavior of the system. The theory for this model is taken from the fluent software.

### B. Specification of Problem

Consider a steady state fluid flowing through a elbow of pipe of constant cross section. A 2D model of elbow is as shown in Fig. 2. Elbow Diameter are taken as ½", ¾" and 1" respectively. The inlet velocity is u (m/s), which is constant over the inlet cross-section are taken to be 0.75 m/s, 1.0 m/s, 1.5 m/s, 3 m/s. for Elbow. The fluid flowing through pipe is water at ambient atmosphere which is at a pressure of 1 atm.

### C. Geometry creation in Gambit -

To build a model Elbow modeling software Gambit 2.1 is used. This is user friendly for creating geometry also for meshing purpose.

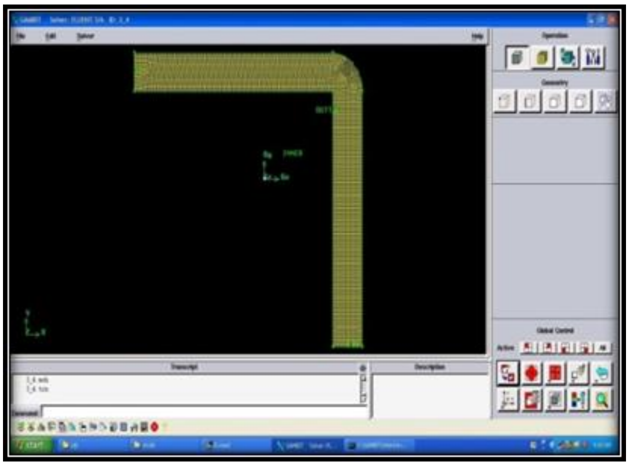


Fig. 1: Geometry of Elbow Created in Gambit

## IV. CFD ANALYSIS

The commercial CFD code and mesh generation packages Fluent 6.3 and Gambit 2.01 were used respectively. Two dimensional steady state Reynolds Averaged Navier Stokes (RANS) equations were solved using the segregated implicit solver. The right choice of a turbulence model is a critical when an industrial turbulent flow problem is faced, especially when this problem involves two dimensional flow phenomena, which need an accurate modeling. It is known that no existing turbulence model is suitable for all flow situations and one numerical setup that yields high accuracy in simulation of one fitting may lead to a large error in that of another. Therefore, CFD validation should be performed. The RANS equations were computed using the k- ε model and Reynolds stress model (RSM). Although the k- ε model is robust, efficient and very widely used, it is known that in highly swirling flows or in flows where significant stream curvature exists, this model becomes inadequate. In such cases, the RSM generally offers greater accuracy by modeling the Reynolds stresses directly. The relative advantages of the two models in simulating Elbow will be discussed in the following section. Non equilibrium wall functions were used for the treatment of the near wall layer. Because of the capability to partly account for the effects of pressure gradients and departure from equilibrium, the non-equilibrium wall functions are recommended for use in the complex flows involving separation, reattachment, and impingement where the mean flow and turbulence are subjected to severe pressure gradients and change rapidly. The second order scheme was used for the RANS equations calculations, with a pressure velocity coupling achieved using SIMPLEC algorithm. The default under relaxation factors were used to aid convergence for all models.

Mesh resolution was driven by the wall y+, and considerable care was taken to ensure that the aspect ratio of the cells was as uniform as possible to the general features of the flow. The expansion ratio was generally kept below 1.2 to ensure sufficient mesh refinement throughout the domain. In accordance with the requirements of the non equilibrium wall function, the value of y+ was set to be between 30 and 300 for the majority of the calculations, and the expansion ratio was set to be 1.1. A variety of grid densities was tested to ensure that the grid is sufficiently dense for the accurate representation of the possible large gradients of flow variables and the overall grid size is as small as accuracy can allow for reasonable convergence speed. The doubling of grid density was not implemented simultaneously in all the two dimensions and sections of the computational domain. This arrangement enables the grid dependency tests to be carried out without dramatically increasing demands on computing resources. Following figures shows uniform structured cells near the walls for the Elbow, meshes consisted of 3229 quadrilateral cells, which approach the limit that can be handled with the computing resources available. These were specified the velocity at the inlet and the Pressure at the outlet as boundary conditions for the pipe Elbow

½” Elbow – Inlet Velocity 0.75 m/s

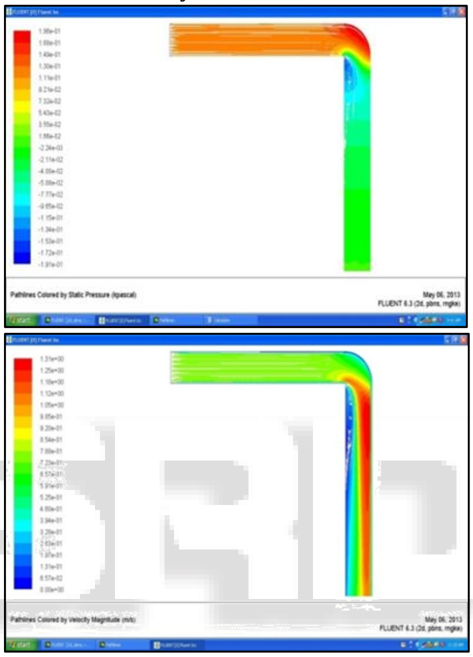


Fig. 2: CFD Analysis of fluid flow through ½” Elbow for 0.75m/sec velocity

½” Elbow – Inlet Velocity 1.0 m/s

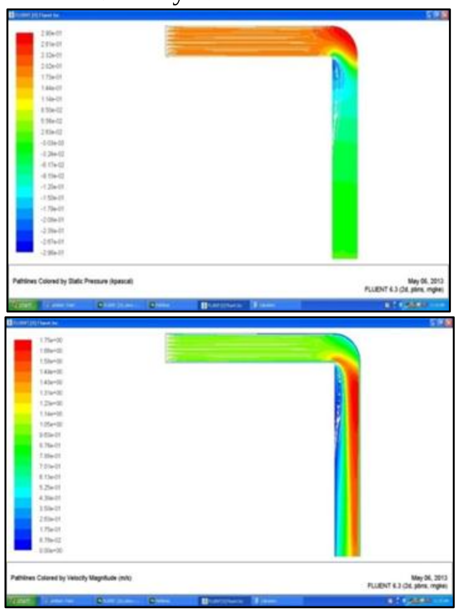


Fig. 3: CFD Analysis of fluid flow through ½” Elbow for 1m/sec velocity

½” Elbow – Inlet Velocity 1.5 m/s

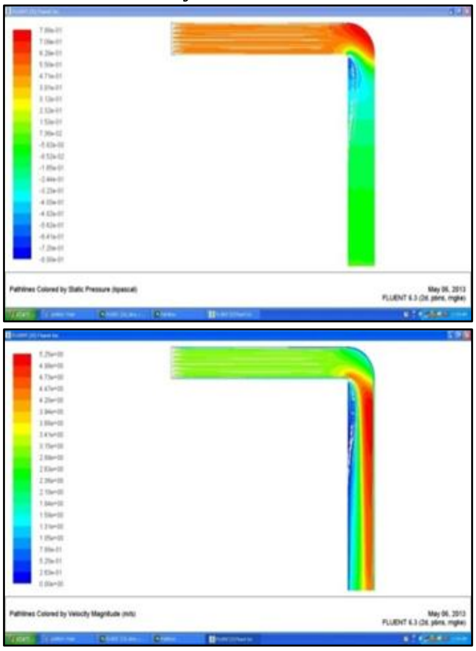


Fig. 4: CFD Analysis of fluid flow through ½” Elbow for 1.5m/sec velocity

½” Elbow – Inlet Velocity 3m/s

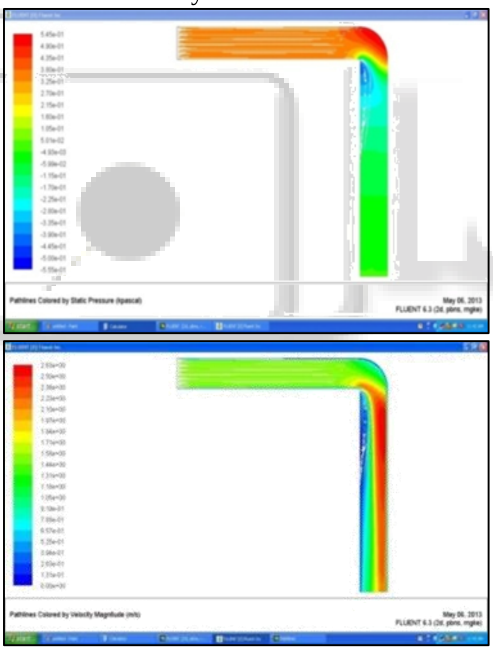
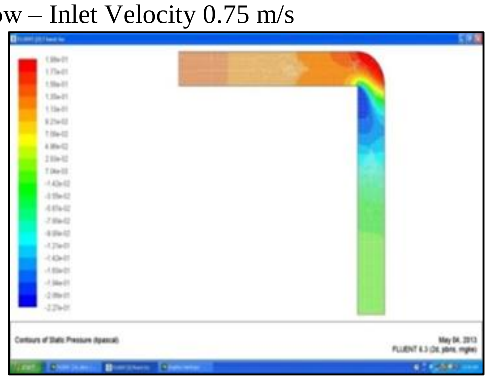


Fig. 5: CFD Analysis of fluid flow through ½” Elbow for 3m/sec velocity

1” Elbow – Inlet Velocity 0.75 m/s



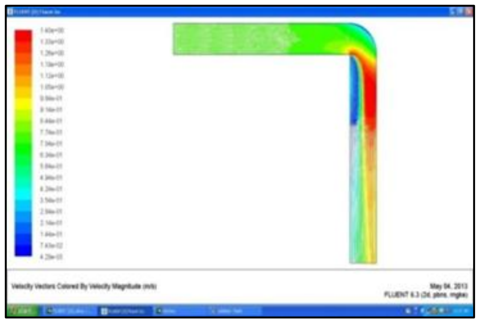


Fig. 6: CFD Analysis of fluid flow through 1” Elbow for 0.75m/sec velocity

1” Elbow – Inlet Velocity 1.0 m/s

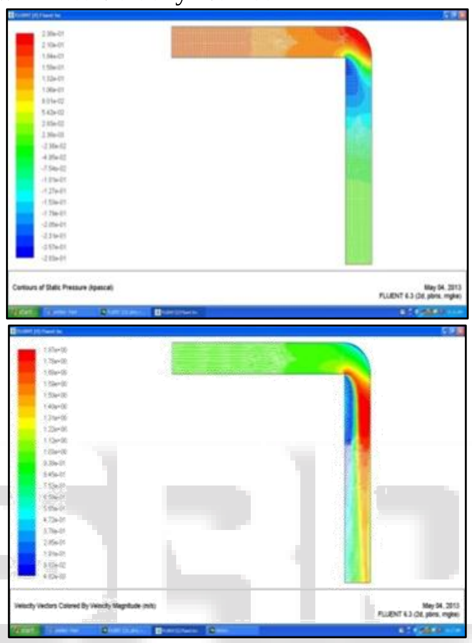


Fig. 7: CFD Analysis of fluid flow through 1” Elbow for 1m/sec velocity

1” Elbow – Inlet Velocity 1.5 m/s

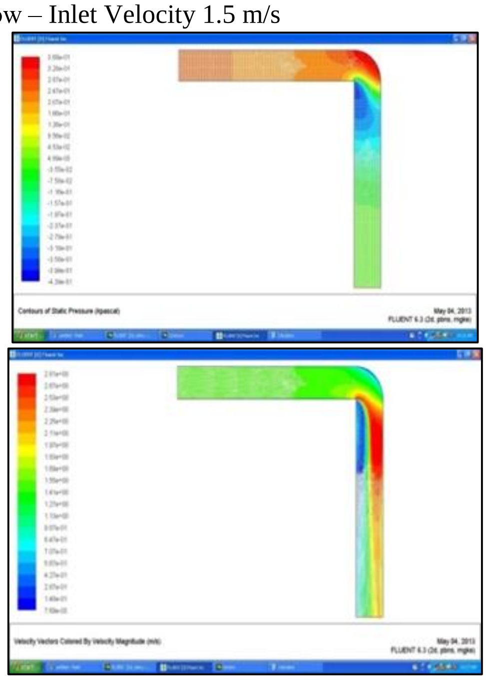


Fig. 8: CFD Analysis of fluid flow through 1” Elbow for 1.5m/sec velocity

1" Elbow – Inlet Velocity 3.0 m/s

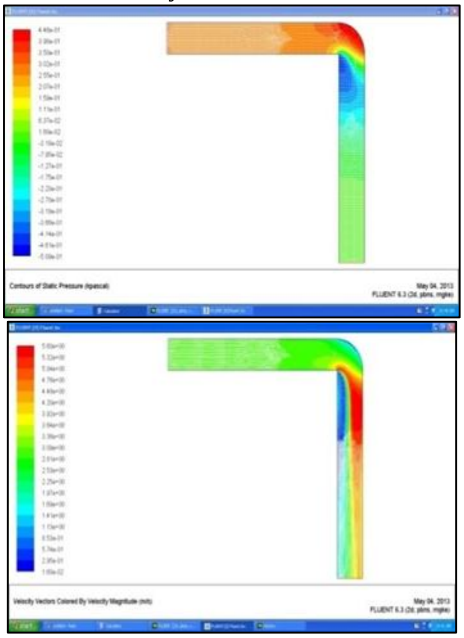


Fig. 9: CFD Analysis of fluid flow through 1" Elbow for 3m/sec velocity

## V. RESULT AND DISCUSSION

When the fluid enters an elbow, a pressure gradient is set up to provide the necessary inward acceleration. If the flow has a non-uniform velocity, the pressure gradient is insufficient for the faster moving particles and more than sufficient for the slower ones. The elbow has a constant radius and the pipe a constant cross-section and is sufficiently long, the curved flow eventually becomes fully developed, and i.e. the velocities do not vary with distance along the pipe axis.

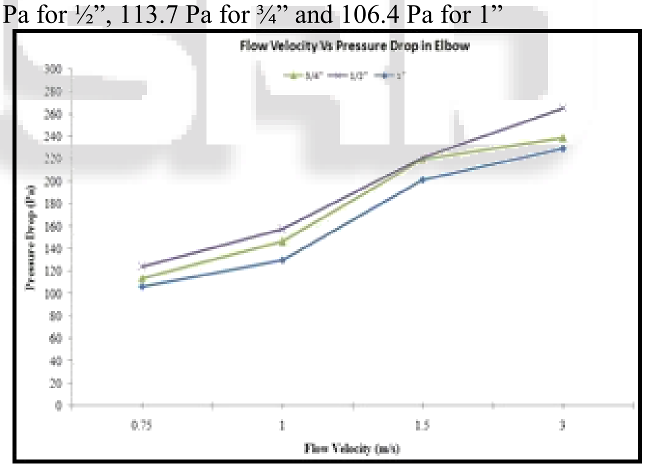
The contours of static pressure and velocity magnitude in central symmetry plane are presented for each of the three turbulence models. The pressure is seen to be transversely uniform at the far upstream and downstream in the straight pipe region. A region of high pressure occurs at the outer wall of the elbow as the flow decelerates, and almost the whole round elbow is covered with the high pressure with both k- ε, whereas, only halfway of the elbow is in high pressure by RNG. A region of low pressure is formed at the inner wall as the flow accelerates around the elbow, and the low pressure region shows a further extension and more influence on the downstream flow than the both k- ε model. The velocity contour indicates that a larger low velocity zone which locates at the downstream from the separation point. The flow attached to the wall and the pressure recovers to uniform with a constant axial gradient at approximately 4.40d downstream of the Elbow for the RNG k- ε model. The primary flow downstream is more closed to the inner wall for RNG k- ε model

The pressure variations along the inner and outer walls of the Elbow in central symmetry plane with different Re and d. The assumption is made during the computation that the water velocity distribution at the inlet of the pipe elbow is non uniform and the flow direction is normal to the inlet cross section. Because of the fixed pipe diameter, water density and viscosity, the Reynolds number (Re) only depends on the water velocity. For the inner wall, the flow is accelerated into the elbow and the wall experienced a positive pressure gradient up. At further downstream, an adverse pressure gradient presented, causing the local flow separation, For the outer wall, on the contrary, an adverse pressure gradient presented first, and then a positive pressure gradient, leads to the results that the pressure gradient changes rapidly, especially at inner and outer walls downstream of the elbow and the lower and higher pressure come into being. The pressure gradient and magnitude will depend on Reynolds number Re, and that the transition point or zone of pressure gradient is determined by curvature ratio d. The following table 1 shows the comparative pressure drop between Elbow for Different diameter of pipe and for different inlet velocities.

| Sr. No. | Water velocity at Inlet (m/s) | Pressure Drop | |
|---|---|---|---|
| | | 1/2" Pipe Diameter Elbow | 1" Pipe Diameter Elbow |
| 1 | 0.75 | 124.1 | 106.4 |
| 2 | 1.0 | 157.2 | 129.8 |
| 3 | 1.5 | 221 | 201.4 |
| 4 | 3.0 | 265 | 229 |

Table 1: Comparative of Pressure drop between Elbow

From the Graph 1, it is clear that, in case of elbow as the size of pipe diameter decreases i.e. from 1" to 1/2" and velocity at the inlet increases i.e. from 0.75 m/s to 3 m/s, the pressure drop increases. It is clearly seen that for inlet velocity 0.75 m/s for Elbow a pressure drop is found to 124.1 Pa for ½", 113.7 Pa for ¾" and 106.4 Pa for 1"




Graph 1: Flow velocity Vs Pressure Drop in Elbow

## VI. CONCLUSION

In analyzing pressure losses through elbow, pipe outlet conditions and inlet velocity profile are important. Loss coefficients depend on some parameter which varies with the phase of the circulation at the elbow outlet, Also there is a contribution to the loss in the downstream transition region of a given system not only associated with the value of the flow circulation at elbow outlet but also the displacement of the fluid particles there.

CFD analysis of fluid flow in 12.7mm (½") , and 25.4 mm (1") pipe diameter with r/d ratio of 1.5 and 3 was performed using commercially available CFD code FLUENT. Analysis was performed for four different water velocities i.e. 0.75 m/s, 1.0 m/s, 1.5 m/s, 3.0 m/s, in each of the three elbows. Pressure drop profiles and their respective pressure contour maps were presented for characteristic flow behaviors in fluid flows. .

The pressure gradient and magnitude will change with different Reynolds number Re, the increment of Re will leads to the results that the pressure gradient changes rapidly, especially at inner and outer walls downstream of elbow, and the lower and higher pressure come into being. The transition point or zone of pressure gradient is related with curvature ratio. As the curvature ratio decreases gradually, the transition point of pressure gradient moves towards the inlet of the pipe fitting for the outer wall but keeps almost unchanged up to an enough larger curvature for the inner wall. The flow, which adds significantly to the pressure loss, causing a rapid rise in pressure loss factor k as the elbow tightens. The above conclusions and data refer specifically to the water flow in the elbow with higher Re and may necessarily apply to only steel pipe fittings. An appreciation for the accuracy of the methods being employed enables the engineer to achieve a safe and economical design.

## References

[1] ABHAI KUMAR VERMA and S.N. SINGH, " Pressure drop for the flow of high concentration solid liquid mixture across 900 bend" Indian Journal of engineering & material science vol. 13 pp. 477-483, 2006.
[2] RABIN DEBNATH, "Experimental Investigation of Turbulent Fluid Flow through a Rectangular Elbow". International Journal of Engineering Science and Technology Vol. 2(6), 1500-1506, 2008
[3] AKINTOLA &O. GIWA, "Optimum Pipe Size Selection for Turbulent Flow" Leonardo Journal of Science,ISSN 1583-0233, 2009.
[4] BOHUSLAVKILKOVSKY&ZDENEKJEGLA, "Comparison of Different Methods for Pressure DropCalculation in 90° and 180° Elbows" Institute of Process and Environmental Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technicka, 616 69 Brno, Czech, 2011
[5] T. K. BANDYOPADHYAY and S. K. DAS " Non-Newtonian and Gas-non–Newtonian Liquid Flow through Elbows" Journal of Applied Fluid Mechanics, Vol. 6, No. 1, pp. 131-141,2011.
[6] QUAMRUL H. MAZUMDER, "CFD Analysis of Single and Multiphase Flow Characteristics in Elbow" Engineering, 2012, 4, 210-214 doi:10.4236/eng, 2012.