Sirte University Scientific Journal
Vol. 4, No. 1, June 2014

3D Fluid flow in an Elbow Meter-CFD Model

Abdalsalam M. Muftah

Faculty of Engineering, Sirte University, Libya
e-mail: salamgader@su.edu.ly

Abstract

Elbow meter is one of the common flow measurement systems which are used to determine the pressure difference occurring as a fluid flows change by resistance. This differential pressure exists when a flowing changes direction due to a pipe turn. The pressure difference results from the centrifugal force. Since pipe elbows exists in plants and its cost is very low. However, the accuracy is very poor [2]. For this reason, the purpose of this paper is to run a CFD Model (Computational Fluid Dynamics) at Elbow meter using Solidworks Flow Simulation [1] with different pipe sizes, ranging from $6''$ *to* $12''$ *nominal diameter. The CFD Simulation will be extended to run with using different fluid types, varying as air, steam, oil and water. The goal is to determine the sensitivity of flow measurements in regards to these parameters. CFD results will be compared to a corresponding theoretical solution to investigate how much the accuracy can be improved by changing the geometry of elbow pipe and fluid type.*

Key words: *Computational Fluid Dynamics (CFD), Elbow meter, Pressure drop*

1. Theoretical Overview

Elbow meter, flow measurement device, is the most widely applied in industrial and laboratory practice. Several investigations have been reported to determine the friction factor and pressure drops in horizontal [3] and vertical [4]. The principal of operation of this device consists basically in determining the pressure difference which is measured on points at inner and outer side of elbow duct. Sometimes, instead of preparing characteristic of the device, simple algebraic relations are used derived from experimental data, it has been points at inner and outer side of elbow duct. Sometimes, instead of preparing characteristic of the device, simple algebraic relations are used derived from experimental data, it has been shown that Bernoulli's equation can be modified to relate the pressure and elevation at these pressure points by introducing a bend coefficient term C_k which varies from 1.3 to 3.2 depending on the geometry of the elbow [5].

$$C_k \frac{V^2}{2g} = \left(\frac{P_O - P_i}{\gamma}\right) + (Z_O - Z_i) \quad (1)$$

By setting all terms equal to V, we get

$$V = \frac{1}{\sqrt{C_k}}\sqrt{2g\left(\frac{P_O - P_i}{\gamma}\right) + (Z_O - Z_i)} \quad (2)$$

Then substituting this equation into the relation $Q = V * A$, we get

$$Q = \frac{A}{\sqrt{C_k}}\sqrt{2g\left(\frac{P_O - P_i}{\gamma}\right) + (Z_O - Z_i)} \quad (3)$$

The pressure difference (dP) can be determined by setting all terms equal to $P_O - P_i$

$$dP = \gamma\left\{\frac{Q^2 * C_k}{2gA^2} - dZ\right\} \quad (4)$$

Where $dZ = Z_O - Z_i$ and $Q = \frac{\dot{m}}{\gamma}$

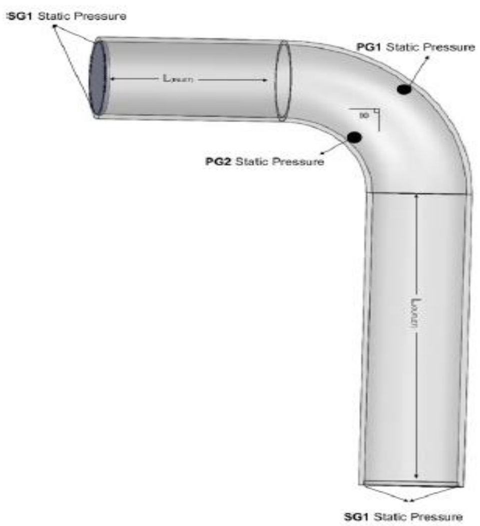


Figure 1 Elbow Meter Geometric

The pressure difference (Δp) and velocity (V) in CFD simulation can be calculated by the following equations:

$$\Delta p = \frac{Q^2 \rho D}{r A^2} \quad (5)$$

$$V = \sqrt{\frac{r \Delta p}{\rho D}} \quad (6)$$

Where r is the radius of curvature, D is pipe diameter and ρ is the flow density

2. CFD Model Geometry

Elbows utilized on the CFD Simulation have an angle of curvature of 90^o , average curvature and nominal diameter are ranging from $6'' \; to \; 12''$. in general, the curvature pipe section length was equal to 1.5 times the nominal diameter, the straight pipe section length on the inflow side was given a length of two times of nominal diameter, while the straight pipe section length on the outflow side was given a length of four times the nominal diameter. Static pressure sensors were located at inner and outer of curvature section shown in Figure (1). The described elbows are characterized by a high level of smoothness both on inner and outer surfaces.

The other CFD Flow parameters as following:

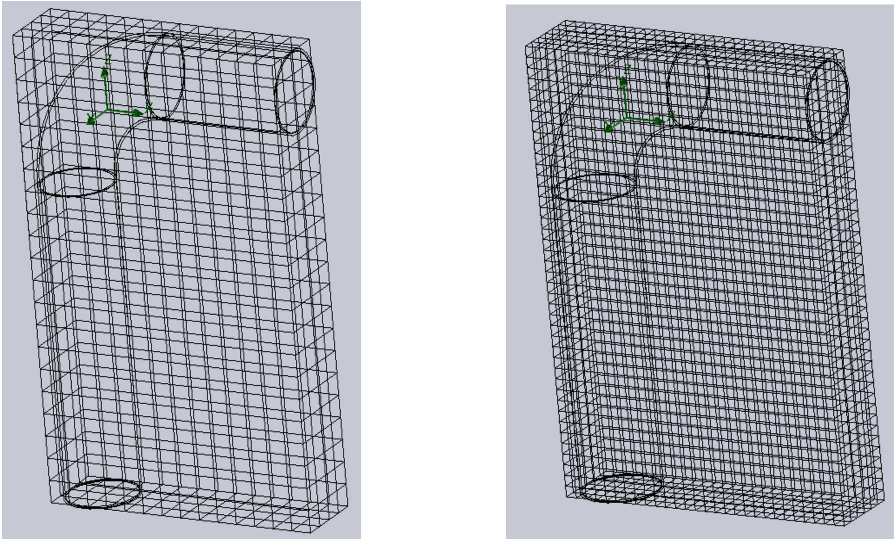
Inlet Mass flow 1.0 lb/sec , Environment Pressure∼14.7 Psi,

Fluid Types are air, steam, oil and water. The fluids have a constant temperature at $68F^{\,O}$

The flow has no inlet velocity but it was effected by Gravity only which equals to $32.2 ft/sec^2$

3. The CFD Meshes and Convergence

The CFD Models were run successfully for all cases. The initial mesh consists of 1484×1664 cells as shown in figure (2-a). For simplicity and computational time, the mesh was initially settled at level 3. The run was converged at small computation time around 18 second and iteration equals to 58. Since our objective to improve the accuracy to CFD Models, the meshes were refined until level 6 as shown in figure (2-b). Increasing the refining level more than level 6 never gave any improvement for the systems. A summary of these results is listed in Table 1.



a- Initial mesh b- Final mesh

Figure 2 CFD Mesh, a) initial mesh, b) final mesh

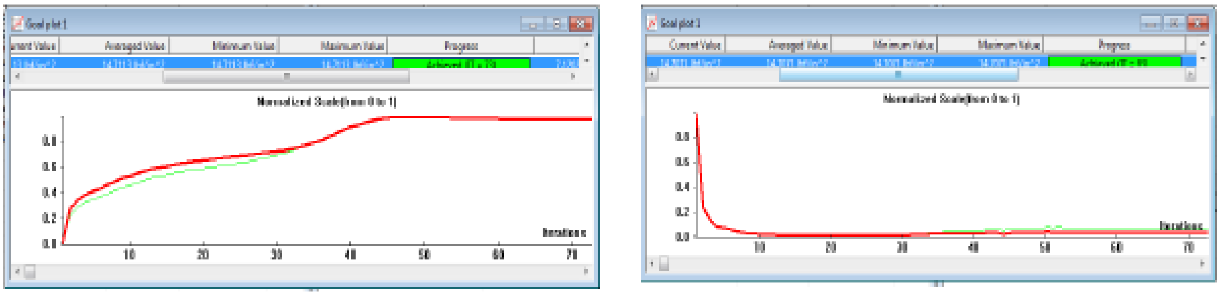


Figure 3 Samples of CFD Model Convergence

Table 1. The Refined Mesh

Elbow size	Refined mesh	Iteration
6 inch	12756×7496	101
8 inch	11872×6248	119
10 inch	11894×6248	83
12 inch	11881×6248	118

4. CFD Results

This study has been conducted on four CFD specimens with different diameter sizes varying from 6 to 12 inch, and on four CFD specimens with different liquid viscosities such as air, gas, water and oil.

4.1 CFD Model with diameter size variation:

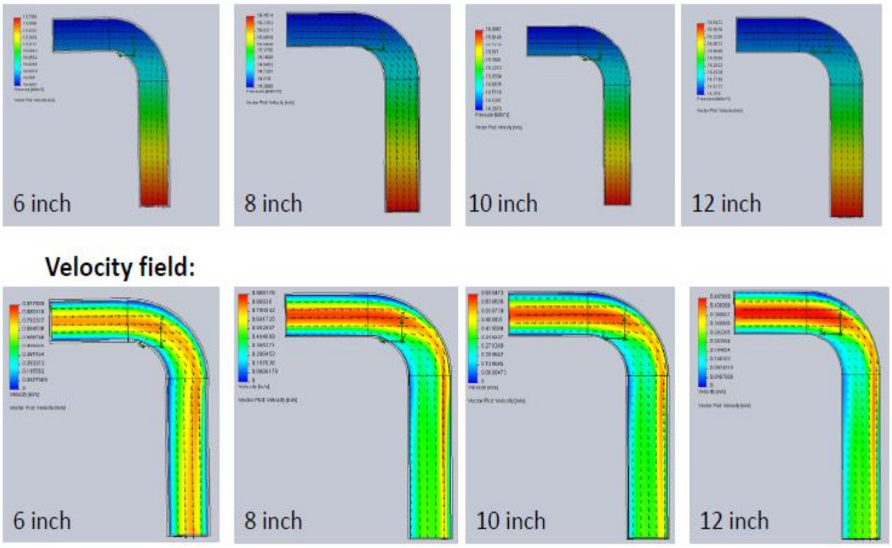


Figure 4 CFD Results a) Pressure distribution b) Velocity field

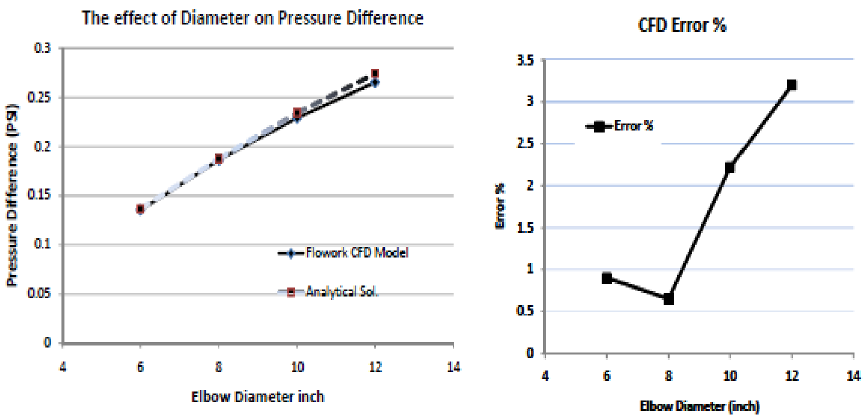


Figure 5- a) Comparison of pressure difference in CFD Models with analytical solution b) The effect of pipe diameter on CFD Error

Table 2 The sensitivity of Pressure difference to Elbow diameter size

Diameter (inch)	Pressure Difference (CFD Model)	Pressure Difference (Analytical Solution)	Error %
6	0.1357	0.1370	0.9 %
8	0.1863	0.1875	0.65 %
10	0.2291	0.2343	2.22 %
12	0.2652	0.2740	3.2 %

4.1 CFD Model with fluid property variation:

Table 3 The sensitivity of Pressure difference to flow property variations

Fluid Type	Pressure Difference (CFD Model)	Pressure Difference (Analytical Solution)	Error %
Air	0.005842	0.0065	10.12 %
Steam	0.011304	0.0119	5.00 %
Oil	0.210691	0.2145	1.77 %
Water	0.22906	0.2343	2.23 %

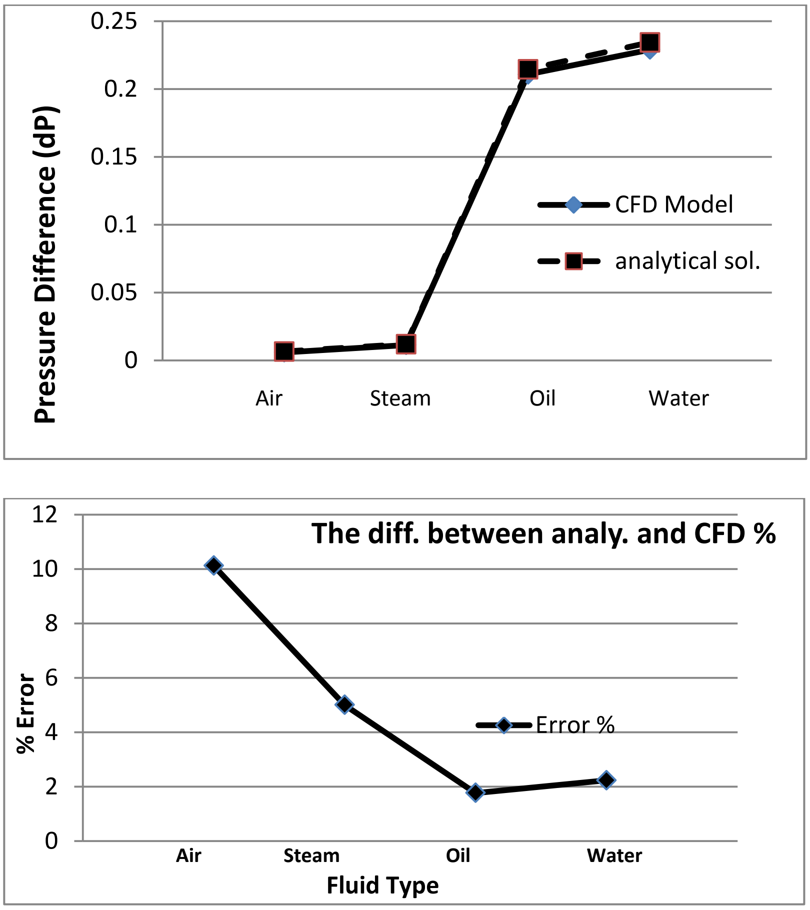


Figure 6- a) Comparison of pressure difference in CFD Models with analytical solution b) The effect of fluid type on CFD Error

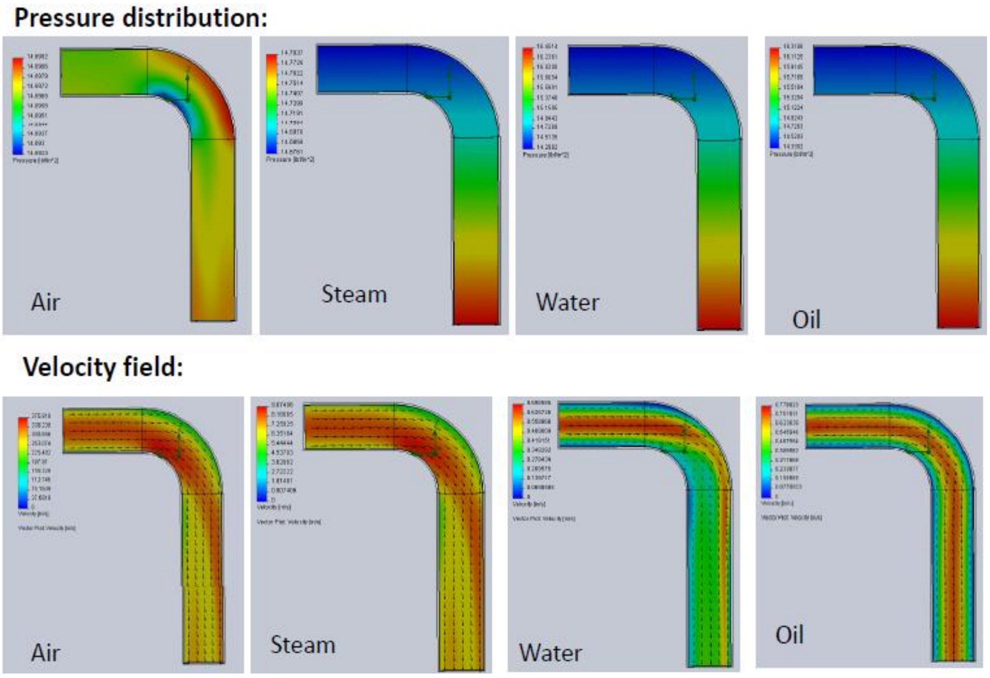


Figure 7 CFD Results a) Pressure distribution b) Velocity field

5. Discussion and Conclusion

The paper was carried out of two stages. First one was to study and investigate the sensitivity of pressure difference and distribution to the change in Elbow geometry. The results show that pressure difference increased with increasing in pipe diameter. In this paper, the error between the analytical solution and CFD outputs ranged from 0.9% to 3.2%, the larger pipe size giving larger errors. The high accuracy in present model may due to using high level of refining CFD mesh.

The second stage of this paper was to study the sensitivity of pressure difference to changing of fluid type. The results show that the pressure difference increase as the fluid transmit from gas to liquid phase. The error between the analytical solution and CFD outputs ranged from 1.77% at oil to 10.12% at air. The results show that errors in Elbow meter was not depend on pipe geometric only but it also depend on the fluid which might be used. The results also recommend that in industries which used steam and liquid flow, elbow meter can give a good accuracy according to this study.

In general, the analytical method is general, systematic and significantly more accurate than computer simulations. Although Bernoulli's equation which basically used to measure the pressure loss in this paper is a simple algebraic relation, its results were not that quite consistent. For example, the result does depend on the bend coefficient term C_k which varies from 1.3 to 3.2 depending on the geometry of the pipe. This coefficient needs to be well-tested and reliable measurements are to be made. In addition, the experimental results are at considerable variance with one another in regard the best judgment of value of C_k. Generally speaking, it appears that C_k is considered not convinced especially for large pipe which no information of values has been found [5].

In this paper, it has been used this formula [1] to determine the bend coefficient C_k

$$C_k = \left[0.13 + 1.85\left(\frac{r}{R}\right)\right]\sqrt{\frac{\theta}{180}} \tag{5}$$

R: *Bend Radius*, r: *Pipe Radius*, θ: *Bend Angle*

Although, the effect of bends were also not considered in the CFD simulation, the results has shown a great approximation towards the analytical solution especially when the mesh has been refined. Generally speaking, it is hard to say that the analytical solution is exact solution with the observation on the bend coefficient or to say that CFD Model can give exact solution due to the computational considerations.

References

[1] Mutsson J. E. (2011). An introduction to Solidworks Flow Simulation 2011, SDC Publications

[2] Figliola R. S. and Beasley D. E. (2011). Theory and Design for Mechanical Measurements, Fifth Edition, Wiley Publication.

[3] Cole J. S., Donnelly G. F., and Spedding P. L.(2004). Friction factors in two phase horizontal pipe flow, International Communications in Heat and Mass Transfer, vol. 31, no. 7, pp. 909–917.

[4] Wongwises S. and Kongkiatwanitch W.(2001). Interfacial friction factor in vertical upward gas-liquid annular two-phase flow," International Communications in Heat and Mass Transfer, vol. 28, no. 3, pp. 323–336.

[5] Pritchard P. J. and Leylegain J. C. (2011). Introduction to Fluid Mechanics, Fifth Edition, Wiley Publication.