

Investigation of flow inside sharp bend elbows **using Numerical Fluid Mechanics.**

Monametsi Gadiile and N.Subaschandar

Department of Mathematics and Statistical Sciences,
Botswana international University of Science and Technology
(BIUST), Palapye, Botswana
Monametsi.gadiile@studentmail.biust.ac.bw

Bent pipes and ducts are used extensively in the industrial complexes mainly due to space constraints. However, bends in pipes and ducts cause pressure losses and generate flow separation inside in the pipes and ducts reducing the efficiency of fluid transport. Numerical investigations of turbulent air flows inside sharp bend elbows with different angles of bend were conducted using ANSYS FLUENT the commercially available CFD software. Computational fluid dynamics simulations were performed using FLUENT software with 2D, steady, turbulent and incompressible flow conditions assumed. Standard k- ϵ turbulence model was used to calculate turbulence quantities. The predictions of mean velocity profiles of the air flow inside the sharp 90° elbow duct were compared against the experimental results obtained using laser Doppler Anemometry (LDA) and other available computational studies. Reasonable agreement was obtained. The present computational study was extended to flows inside the bend elbow with bend angle ranging from 0 deg to 150 deg, from the point of view of calculating pressure loss due to the bend and to capture secondary flow characteristics in the elbow region.

Keywords : Bent ducts , CFD , FLUENT , turbulence model

INTRODUCTION

Bends are an important part of any pipeline network system as they provide flexibility in routing or carrying of the fluid through various systems. These pipe bends are used in industries mainly in, among other things, ventilation pipes, heat exchangers and turbine machineries, and even in car engines and many other engineering fields and applications such as, High Temperature Gas Cooled Reactor (HTGR), Pressurized Water Reactor (PWR), CANDU type reactor, torus reactor, oil and gas production field with their distribution networks, heat exchangers, internal combustion engines, solar collectors, the cooling system of processing industries, electronic components and nuclear power plants. Some of these duct systems and multi duct conveying systems consist of not only 90 degree bends (often curved and sharp) but also other angles due to limitations of space. Although pipe bends are unavoidable, they are a source of complex secondary flows, as a result of recirculation zones, losses of energy and pressure, and also variable heat transfer rates which may affect overall performance or present other design constraints. In other instances presence of geometries producing high total pressure losses or heat transfer maybe intentional, but there is need for them to be accompanied by detailed understanding and/or predictive techniques.

One of the main characteristics of fluid flow through pipe bends are the presence of adverse pressure gradient developed by the centrifugal force acting on the flow. Due to the presence of centrifugal

force and pressure gradient, the fluid moves towards the outer side of the bend and comes back towards the inner side. For a tough bend curvature, this adverse pressure gradient near the inner wall may start the flow separation developing a secondary flow allowing a large increase in pressure loss. This increase of pressure loss experienced in the pipe bends are generated by friction and momentum exchanges appearing from the change of flow direction. Reynolds number, bend curvature ratio and bend angle are the important factors. Therefore investigations of flows through these pipe bends are of great significance in understanding and improvement of their performance and minimization of the losses in pressure and energy especially. Even more interesting is that curved elbows cause less energy loss than sharp elbows but their production is more costly than sharp elbow bends.

Energy loss in sharp elbows is caused by formation of separation zones, which appears just after the corner and stability of separation, is questionable, and therefore numerical prediction of flow in sharp elbow becomes difficult. Hence the need which will be undertaken in the present study, to study other possible ways that can help address the problems of high energy loss, reduce flow separation and secondary flow effects on fluid flow through sharp bends by investigating the effects of redirecting fluids sharply using various angles of 30 degree multiples starting with the 30 degree angle to the 120 degree angle bend.

Flow pattern in 90 degree pipe bends with different curvature ratio and different Reynold's number has been studied by P. Dutta and M. Nandi [1]. It was reported that the normalized mean velocity profile has a low dependency on Reynold's number for low curvature ratio and for high curvature ratio it tends to recover its fully developed shape while Reynold's number is increasing. It was also discovered that swirl intensity has a weak dependence on Reynolds number and high dependence on curvature ratio as also witnessed by Kim et al., [2]. Flow separation in common 90 degree bends was studied by numerical methods based on CFD by Dutta et al., [3], and it was discovered that starting point of flow separation moves upstream in the bend as Reynolds number increases, while reattachment moves downstream, this trends corresponds to the dependence of Reynolds number on total pressure loss, as also seen by Tanaka et al., [4] and is consistent with experimental results of Takamura et al, and Kimura et al.[5, 6]. 2-D analysis of Turbulent fluid flow of forced convective heat transfer in elbow duct of rectangular cross-section, with uniform wall temperature conditions and different inlet uniform velocities of fluid flow was done by Debnath et al., [7]. It was seen that there was generation of recirculation at different bend positions. The recirculation was however strongest at the upper limb of the duct. It was also realized that secondary flow of recirculation strongly influences the main stream flow as well as the heat transfer phenomena. Three dimensional single phase turbulent flow within 90° pipe bend was studied by computational fluid dynamics method using k- ϵ turbulent model by Dutta et al., [8]. Experimental data of Sudo et al.,[9] was successfully applied to validate the model used. Reasonable agreement was obtained. It was observed that, for increase in Reynolds number, the flow recovers its fully developed velocity profile. This observation is also consistent with experiments carried out by Kim et al, [2], and Zagarola, and Smits [10]. A reverse flow was observed due to the adverse pressure gradient at the outlet of the

bend where the momentum is lower than that near free stream, by which the velocity near the wall reduces and the boundary layer thickens[8].

PROBLEM DEFINITION.

The main objective of the present study is to investigate the effects of multiples of 30° angle pipe bends [ranging from 0° (a straight duct) to 120°] on a turbulent single phase fluid flow. The study will mainly be focused on pressure loss caused by these types of sharp bends, the length of the separation zones and the effect of Reynolds number on the separation length. Pressure loss on a 90° sharp bend will be studied. A comparison of the performance of the two turbulence models K-omega and k-epsilon will also be made. However, some preliminary results are only presented in this paper.

MATHEMATICAL EQUATIONS

In this section , we present the governing equations of the fluid flow inside the bent ducts are presented. The flow is assumed to turbulent, steady, incompressible and two-dimensional. Hence the governing equations are the Reynolds Averaged Navier-Stokes equations describing the mean flow distributions inside the duct. These equations are given below :

Momentum Equation in x-direction

$$\frac{\partial}{\partial x}(\rho \bar{u} \bar{u}) + \frac{\partial}{\partial y}(\rho \bar{v} \bar{u}) - \frac{\partial \bar{p}}{\partial x} + \mu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} \right) - \frac{\partial}{\partial x}(\rho \bar{u}' \bar{u}') \quad (1)$$

Momentum Equation in y-direction

$$\frac{\partial}{\partial x}(\rho \bar{u} \bar{v}) + \frac{\partial}{\partial y}(\rho \bar{v} \bar{v}) = -\frac{\partial \bar{p}}{\partial y} + \mu \left(\frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} \right) - \frac{\partial}{\partial x}(\rho \bar{v}' \bar{u}') \quad (2)$$

Continuity Equation

$$\frac{\partial}{\partial x}(\rho \bar{u}) + \frac{\partial}{\partial y}(\rho \bar{v}) = 0 \quad (3)$$

Since the flow is assumed to be turbulent the k-ε turbulence model has been used to capture turbulence quantities. The k-ε turbulence model equations are given here. For this model the transport equation for $\bar{k}$ is derived from the exact equation, but the transport for $\bar{\epsilon}$ was obtained using physical reasoning and is therefore similar to the mathematically derived transport equation of $\bar{k}$, but is not exact. The turbulent kinetic energy $\bar{k}$, and its rate of dissipation $\bar{\epsilon}$, for this model are obtained by the following equations.

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + \rho \epsilon + S_k \quad (4)$$

$$\frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (5)$$

Where G_k represents the generation of turbulent kinetic energy the arises due to mean velocity gradients. S_ϵ and S_k are source terms defined by the user.

$C_{1\epsilon}$, $C_{2\epsilon}$ and C_μ are constants that have been determined experimentally and are taken to have following values;

$$C_{1\epsilon} = 1.44, C_{2\epsilon} = 1.92, C_\mu = 0.09$$

σ_k are σ_ϵ turbulent Prandtl numbers for the turbulent kinetic energy and its dissipation rate. These have also been derived experimentally and are defined as follows.

$$\sigma_k = 1.0, \sigma_\epsilon = 1.3$$

The turbulent viscosity at each point is related to the local values of turbulent kinetic energy and its dissipation rate by:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}, G_k = -\rho \bar{u}'_i \bar{u}'_j \frac{\partial u_i}{\partial x_j} \text{ and } S = \sqrt{2 S_{ij} S_{ij}}$$

GRID INDEPEDENCE AND VALIDATION OF OUR WORK.

At the very beginning of our study, our present model and simulation setup are first validated against the existing experimental and numerical data presented in[10,11]. For that intention, same geometrical configuration of [11] is adopted, only difference being mine is a 2-D geometry while theirs is a 3-D, and the velocity profiles of present computations was compared to the velocity profile of [11]. The results in Fig. 1 show reasonably good agreement with both experimental data of [11] and numerical data of [10], thus validating our model and methodology.

Grid independence study was carried out using several grid sizes. The pressure loss is calculated at the end of each simulation. Grid sizes wer varied until the differences between two successive grid sizes was found to be less tan 0.1 %. This grid independence was achieved for 244,708 nodes in the computational domain. Pressure loss between inlet and outlet boundaries was used to arrive at the grid independent results. Hence a grid size of about 245000 nodes was used for all simulations subsequently.

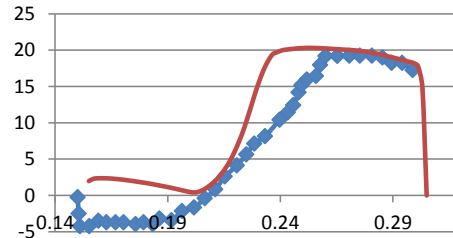


Fig.1 Comparison of velocity profile near the outlet. Symbols with blue line are Experimental data. Brown line is the present computations.

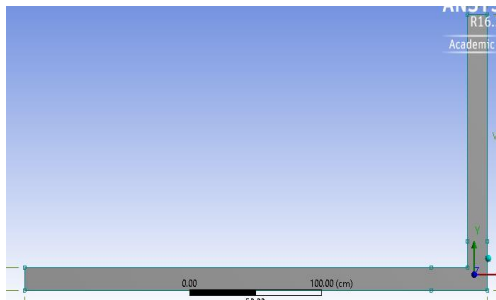


Fig 2 : The geometry with 90 deg sharp bend

Fig. 2 shows the 90° geometry that was used to validate the results and establish grid independence. After validation of the model the current study was extended to other configurations with angles 0°, 30°, 60° and 120° ducts. The duct with bend angle 0° is a straight duct.

RESULTS AND DISCUSSION.

Table 1 below and the Fig 3 below show the pressure drop between the inlet and outlet boundaries of various configurations for both the K-epsilon and k-omega turbulence models.

Now on the Table 1 as in Fig 3 looking specifically at the column for percentage pressure drop between inlet and outlet, it can clearly be seen that the pressure loss increases as the bend angle increases. From the Table 1 and Fig 3 it can be seen that for the straight duct both turbulence models predict the same pressure loss. For lower angle the k-omega turbulence model predicts lower values and for higher angled ducts k-epsilon turbulence models predicts higher values of pressure loss.

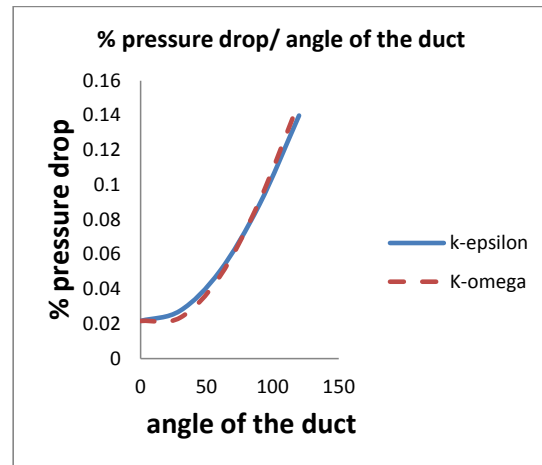


Fig 3 : Pressure loss with duct bend angles for both turbulence models.

Angle of duct	Separation length. (cm)	Separation length/diameter
0°	0	0
30°	0	0
60°	38.38	2.56
90°	71.782	4.782
120°	84.7	5.65

Table 2 : Separation length for various duct bend angles

Angle of the duct	k-epsilon			k-omega		
	Inlet	outlet	% pressure drop	inlet	Outlet	% pressure drop
0	101347	101325	0.0217	101347	101325	0.0217
30	101353	101325	0.0276	101349	101325	0.0236
60	101376	101325	0.0503	101373	101325	0.0473
90	101415	101325	0.0887	101417	101325	0.0907
120	101467	101325	0.1399	101475	101325	0.1478

Table 1: Pressure drop for ducts with various bend angles computed with k-ε and k-ω turbulence models

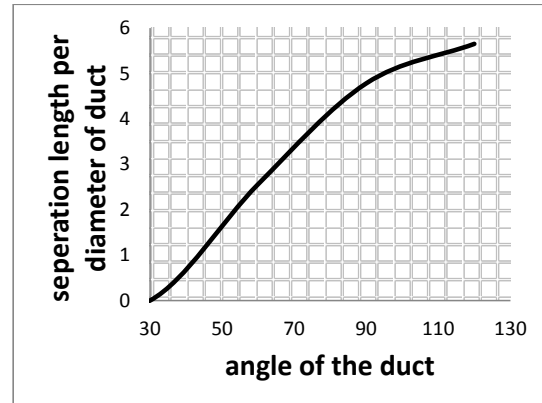


Fig 4 Variation of separation length inside the bend duct for various angles

Now from Table 2, and Fig. 4 above its clear that the separation length inside the duct increases as the duct bend angle increases. It should be noted that the separation length for the straight duct and

30 deg duct are almost zero meaning that there was no flow separation inside both the straight duct and 30 deg bend angle duct.

Velocity [m/s]	Reynolds number [$\times 10^4$]	Percentage pressure drop	Separation length (cm)	Separation length/diameter
5	5.1344	0.02467	66.73	4.45
6.25	6.4180	0.04000	69.63	4.642
7.5	7.7016	0.05228	70.13	4.675
8.75	8.9852	0.06933	71.63	4.775
10	10.26688	0.08874	71.73	4.782

Table 3 : Effect of Reynolds number on pressure loss and separation length

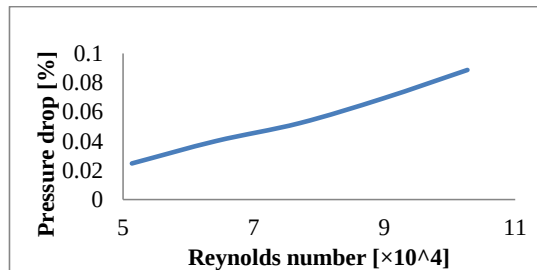


Fig 5 : Variation of Pressure drop with Reynolds number.

Fig 5 shows the variation of pressure loss with Reynolds number. It can be seen from Fig 6 and Table 3 that pressure loss increases as the Reynolds number is increased.

Fig 6 shows the variation of separation length inside the 90 deg bend duct with Reynolds number. It can be seen from Fig 6 and Table 3 that the separation length inside the 90 deg sharp bend duct increases as the Reynolds number is increased.

The present study shows that bent ducts suffer flow efficiency in fluid transport on account of pressure loss as well as the separation zones inside the duct both of which increase as the bend angle increases, thus increasing the performance deterioration of the ducts.

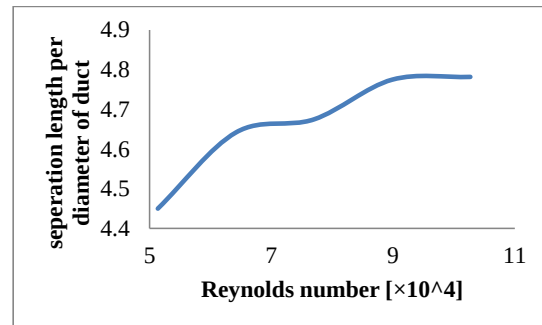


Fig 6 Variation of Separation length with Reynolds number

6. Conclusions

Turbulent flow of single phase incompressible fluid through pipe bends of 30° multiple [ranging from 30° to 120°] has been simulated numerically using k-ε Turbulence modeling in the present study. The validation of 2D models used for the present study with experiments and numerical results reported for 90 deg bend duct indicates a reasonably good agreement. The following conclusions can be made from the present study. It was found that the pressure loss increases with Reynolds number for 90 deg bend duct. Pressure loss increases as the duct bend angle increases. The separation zone size increases inside the 90 deg bend duct increases with increasing Reynolds number. Pressure loss increases with increasing duct bend angle. K-ε turbulence model and k-ω were compared and it was discovered that as far as pressure loss between inlet and outlet of the ducts is concerned, they performed relatively the same.

REFERENCES.

- [1] Prasun Dutta and Nityananda Nandi; Effect of Reynolds number and curvature ratio on a single phase turbulent flow in pipes. 2015..
- [2] Kim, J. Yadav M. and Kim, S. Characteristics of secondary flow induced by 90 degree elbow in turbulent pipe flow, engineering applications of computational fluid mechanics, 8(2), 229-239, 2014
- [3] Prasun Dutta, Sumit Kamur Saha, Nityananda Nandi, Naitri Pal; Numerical study on flow separation in 90 degree pipe bend under high Reynolds number by k-ε modelling, 2015.
- [4] M. A. Tanaka, H. Ohshima, H. Monji; Numerical investigation of flow structure in pipe elbow with large eddy simulation approach, in: ASME 2009 Pressure Vessels and Piping conference, American society of Mechanical Engineers, 2009, pp. 449-458.
- [5] H. Takamura, S. Ebara, H. Hashizume, K. Aizawa, H. Yamano; Flow visualization and frequency characteristics of velocity fluctuations of complex turbulent flow in a short piping under high Reynolds number condition, J. Fluids Eng. 134(10) (2012)m101201.
- [6] A. Ono, N. Kimura, H. Kamide, A. Tobita, Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition, Nucl. Eng. Des. 241 (11) (2011) 4409-4419

- [7] R. Debnath, A. Mandal, S. Majumber, S. Bhattacharjee and D. Roy; Numerical Analysis of Turbulent Fluid Flow and Heat Transfer in a Rectangular Elbow. 2014.
- [8] Prasun Dutta, Sumit Kumar Saha, and Nityananda Nandi. Computational study of turbulent flow in pipe bends, International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 11 (2015)
- [9] K. Sudo, M. Sumida, and H. Hibara; Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend, Experiments in Fluids, vol. 25, no 1, pp 42 -49, 1998.
- [10] M. V. Zagarola, and A. J. Smits, 'Scaling of the mean velocity profile for turbulent pipe flow', Physical review letters, vol. 78 no. 2, pp.239, 1997..