Mechanics and Mechanical Engineering
Vol. 21, No. 2 (2017) 353–361

Change in Flow Separation and Velocity Distribution Due to Effect of Guide Vane Installed in a 90^o Pipe Bend

Sumit Kumar Saha
Nityananda Nandi
Department of Aerospace Engineering and Applied Mechanics
Indian Institute of Engineering Science and Technology
Shibpur, Howrah-711103, W.B., India
sks.iiest@gmail.com
nityananda@mailcity.com

Received (23 January 2017)
Revised (16 March 2017)
Accepted (13 May 2017)

Present paper makes an effort to study the flow separation and velocity distribution for incompressible turbulent flow through 90^o pipe bend due to the effect of guide vane installed in the bend portion. It has been observed here how the normalized velocity distribution profile changes if the guide vane is provided. $k - \varepsilon$ turbulence model has been adopted for simulation purpose. After validating with existing experimental results, a detailed study has been performed for a particular Reynolds number and four different positions of guide vane. The value of Curvature ratio (Rc/D) has been considered as 1 for present study. The results obtained from the present study have been presented in terms of graphical form. A flow separation region was found at bend outlet for flow through 90^o pipe bend without guide vane. This secondary flow separation region was absent for the cases which deals with the flow through 90^o pipe bend with guide vane. Velocity distribution at seven different downstream positions have been presented in graphical form. Position to get a fully developed velocity distribution profile for each cases has been estimated on the basis of presented results.

Keywords: 90^o pipe bend, flow separation, numerical study, velocity distribution, with /without guide vane.

1. Introduction

Study on pipe bends and piping system demands a great importance in modern engineering due to its huge area of applications. Turbulent flow through pipe bends has been an area of interest for researchers from both applied [1] and fundamental view point [2]. Pipe bends are used as a part of several mechanical systems, e.g. heat exchanger [3], reciprocating engines [4] and nuclear reactors [5]. Pipe bends are also used in oil industries, water supply system and also for fluid and material transportation and to change the direction of the flow. So it is important to study about the single phase flow through pipe bends for minimizing the losses by understanding the complex flow behavior and improving their performance. An experimental study on 90^o pipe bend using Laser Doppler velocity meter for Reynolds No 500, 1093, 43000 considering curvature radius as 2.8 times of the pipe diameter was carried out by Enayet et al [6]. They reported that the secondary flow occurs in the inside section of the bend initially. The strength of the secondary flow depends on the inlet boundary layer and an increase in energy production was due to extra strain rates induced by curvature. So and Anwar [7] showed that the mean axial velocity profile at 49D downstream at the bend exit is similar with the measured profile at 18D upstream of the bend curvature. Swirling flow behavior, unsteady nature of secondary vortices, and transition of secondary flow phenomena has been studied by a number of researchers. They investigated the unsteady flow separation, unstable shear layers and Dean Vortices for turbulent flow through 90o pipe bend. Turbulent flows in a 90o pipe bend causes the flow separation near the inner wall which effects in the piping system performance and also can excite strong vibration and flow induced noise. Some researchers have studied structural vibration and noise induced by boundary layer for turbulent flows through different pipe bends [8, 9]. They came up with the improved design of the guide vane to reduce structural vibration, noise and pressure loss in elbows with circular cross section [10–12]. The present work makes an attempt to study influence of guide vane on flow separation and velocity distribution. For this purpose numerical simulations have been carried out for both with and without guide vane conditions. To understand the mechanics in such cases, the change in velocity distributions are presented in graphical form.

2. Numerical methodology

Study on turbulent flow through pipe bends demand a great importance due to its vast engineering application in modern days. The present study has been carried out considering turbulent flow of a single phase, steady, isothermal and incompressible fluid i.e. water through 90^o pipe bend. Three dimensional Reynolds Averaged Navier Stokes (RANS) equations have been solved using a segregated implicit solver to get results. The second order scheme has been used for the calculations, with a pressure velocity coupling achieved using SIMPLE algorithm.

3. Turbulence Model

The governing equations for incompressible fluid flow with constant properties are:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \upsilon \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{2}$$

Equations (1) and (2) are the conservation of mass and momentum respectively,
f_i – vector representing external forces,
υ – is the kinematic viscosity.
Turbulent flows are irregular, random and chaotic. The flow consists of a spectrum of different scales (eddy size) and are basically designed by the fluctuations

of the velocity field. These fluctuations can be very small scale and high frequency. It is very difficult to analyze the flow problem directly. So right choice of turbulent model is necessary to make the virtual model more realistic compare to the experimental model. The turbulence model needs to be selected based on some considerations, e.g. the physics of the flow, the insight into the capabilities and limitations of turbulence models. The $k-\varepsilon$ turbulence model has been implemented for present study purpose because this model achieves better results for single phase flow through pipe bends [13-21]. In this model, the turbulence kinetic energy (k) and the turbulence dissipation rate (ε) are solved to determine the coefficient of turbulent viscosity (μ_t).

Transport equation for k-ε:

$$\frac{\partial(pk)}{\partial t}+\frac{\partial(pku_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon \tag{3}$$

$$\frac{\partial(p\varepsilon)}{\partial t}+\frac{\partial(p\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial k}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{4}$$

Where: u_i represents velocity component in corresponding direction, E_{ij} represents component of rate of deformation, μ_t represents eddy viscosity.

The equations (3) and (4) also consist of some adjustable constants [22], these are as follows:

$C_\mu = 0.09$, $\sigma_k = 1.00$, $\sigma_\varepsilon = 1.30$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$.

4. Problem definition

The present work studies the effect of guide vane installed in the pipe bend on flow separation and velocity distribution. The problem considered here is a single phase flow through 90^o pipe bend, with and without guide vane having inner diameter 0.1 m and curvature ratio (Rc/D) 1. Curvature ratio can be defined as the ratio between the radius of the bend curvature and hydraulic diameter of the pipe. For the sake of brevity, the study is limited to a particular Reynolds number considering four different positions of the guide vane installed in the bend portion. In total five different virtual models (Case1, Case2, Case3, Case4 and Case5) have been considered in the present study to get the desired results. Case1 considers a flow through simple pipe bend (bend without guide vane) whereas case2 to case5 represents bends with guide vanes at different position. Both inlet and outlet length of straight pipe were set up at 10D to save computational time for all the cases mentioned above. Positions of the guide vane has been determined by radius ratios (ratio of the nominal elbow radius to the inside radius of the elbow curvature Ri of the pipe) taken from published literature [23] shown in the Tab. 1. The fluid medium was water having density (ρ) 998.2 kg/m^3 and dynamic viscosity (μ) of 0.001003 kg/ms for the present study with working temperature of 300 K. Three dimensional structured mesh with hexagonal element have been chosen and optimized via a grid independence study. The schematic geometry of the bend and computational mesh are shown in the Fig. 1 and Fig. 2 respectively.

Table 1 Different positions of guide vane installed in the elbow

Case No.	Position of Guide Vane
Case1(Simple)	0
Case2	0.175
Case3	0.308
Case4	0.493
Case5	0.738

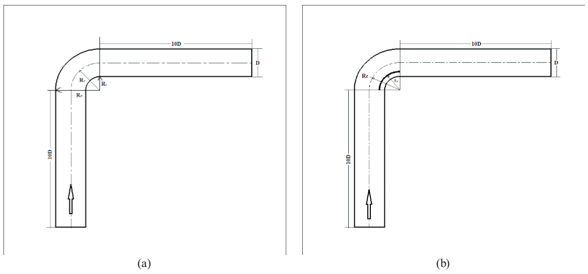

Figure 1 Schematic Diagram of the bend geometry: a) Without guide vane, b) With guide vane

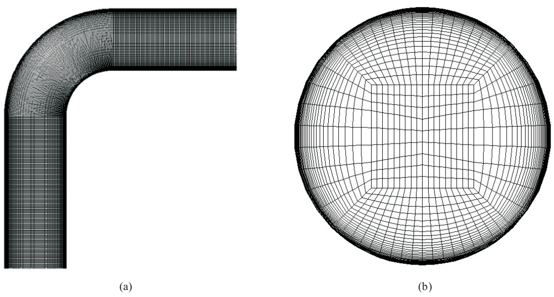

Figure 2 Computational grid of the geometry

5. Validation

Since the current study primary deal with computer based simulation of real life flow process, the validation of computational models are extremely important. Thus a virtual computational model has been formed and compared with published experimental data. Same geometrical configuration and boundary conditions as adopted by Kim et. al. has been used and results obtained from the simulations have been compared against the published experimental data [13]. The authors of the previously mentioned, carried out their studies using a circular cross-sectioned 90^o pipe bend with curvature ratios (Rc/D) of 1.5 for their experimental study for Reynolds number 6×10^4. The present study has been carried out on a computational mesh containing 3.5 million hexahedron elements, which was optimized via a grid independence study. Fig. 3 represents the graphical comparison between the published results and results obtained from present study. Normalized axial velocity distribution at bend outlet position has been compared with Kim et al [13], see Fig. 3.

From Fig. 3, it is seen that the present model is in close approximation with the published data. So it may concluded that the procedure of mesh generation and simulation set up which has been used for validation purpose can be used for further analysis.

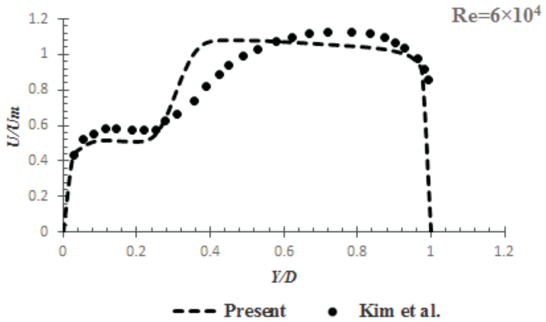

Figure 3 Comparison between Normalized axial velocity profiles of present analysis with published experimental results

6. Results and discussion

The present study has been carried out for a single phase turbulent flow through 90^o pipe bend with/without guide vane conditions. The objective of the present work is to study about the effect of guide vane on flow separation and velocity distribution for a particular Reynolds number and four different position of guide vane. Seven different positions including the bend outlet position along the symmetry plane have been preferred to study about the change in flow separation and velocity distribution for different cases.

Fig. 4 to Fig. 8 represents the normalized velocity distribution at seven different position on the downstream at symmetry plane, for a particular Reynolds number 1×10^5.

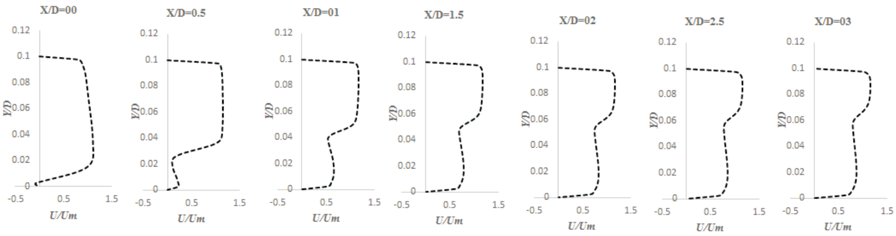

Figure 4 Normalized velocity distribution at different position for case1

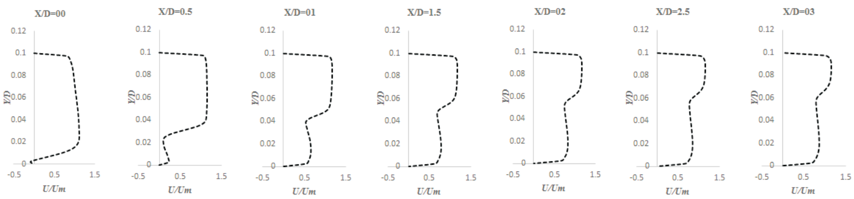

Figure 5 Normalized velocity distribution at different position for case2

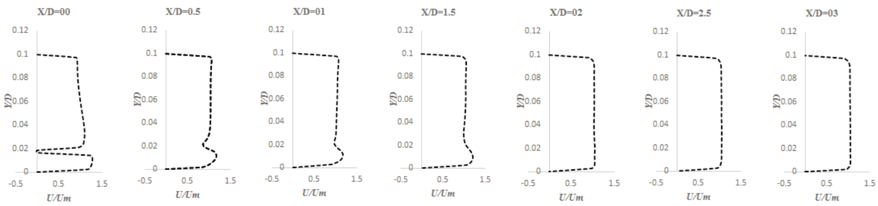

Figure 6 Normalized velocity distribution at different position for case3

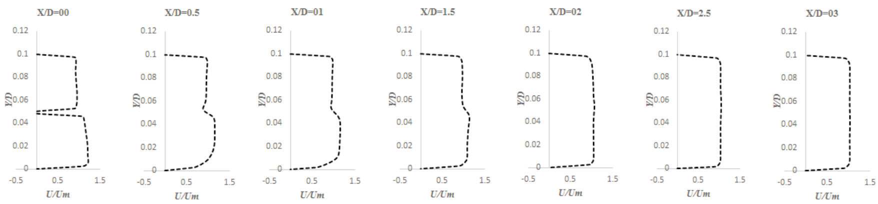

Figure 7 Normalized velocity distribution at different position for case4

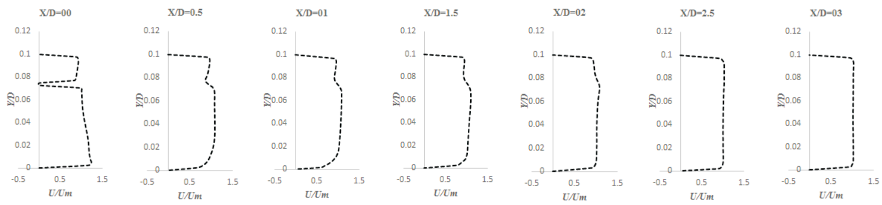

Figure 8 Normalized velocity distribution at different position for case5

The seven plots in each figure describes the pattern of velocity distribution at different positions. The x-axis of the plots represents the velocity distribution normalized by the mean velocity and y-axis represents the non-dimensionalized length of the pipe diameter. $Y/D = 0.5$ represents the centre line position of the pipe at symmetry plane, so min value of $Y/D < 0.5$ refers the inner core side and $Y/D > 0.5$ value refers the outer core side of the pipe.

6.1. *About flow separation*

Normalized velocity distribution at bend outlet position i.e. $X/D = 00$, (Fig. 4) shows a flow separation region for case1. This flow separation region was not synchronous for the other four cases. So it is clear that the guide vane effects on flow and may be the actual cause for non-appearance of the flow separation region for the last four cases dealing with guide vane. Whenever a fluid moves through a curved pipe or pipe bend, there was a radial pressure gradient developed due to the centrifugal force acting on the fluid element. So pressure will be greater at outer core and least at inner core of the bend. The faster moving fluid particle near the central plane and inner core of the pipe have a tendency to move towards the outer core. Therefore an adverse pressure gradient is encountered at the inner core side of bend outlet and flow separation may occur. On the other hand, guide vane split the flow at bend inlet position for the cases 2 to 5. So the radial pressure gradient which mainly developed due to the effect of centrifugal force may not transferal towards the outer core beyond the guide vane and make the flow separated.

6.2. *About velocity distribution*

Each plot in Figs. 4–8 represents the normalized velocity distribution at symmetry plane of the downstream portion. Total seven position from bend outlet has been chosen for each case maintaining equal distance starting from bend outlet position to $X/D = 3$. The term X/D define the position of the collected results at downstream for the present study. The value of X/D represents the length equal to the X/D times of hydraulic diameter. From the fig 4 for case1, it has been observed that the velocity distribution profile did not achieve its fully developed shape up to non dimesionalized distance $X/D = 3$. On the other hand, for case 5 the flow attained its fully developed shape between X/D 2 to 2.5.

For rest of the three cases (case 2,3 & 4) it has been observed that the velocity distribution profile regain its fully developed shape between X/D value 1.5 to 2 though the Reynolds number for all the above mentioned cases are same.

7. Conclusions

Single phase incompressible turbulent flow through 90^o pipe bend for both with- & without- guide vane has been studied numerically using k-ε turbulence model. Change in flow separation and velocity distribution due to the effect of guide vane has been presented. The conclusions obtained from present study may be summarized as follows:

1. The flow separation region has been found for case1 (without guide vane) only at bend outlet, whereas this separation region is not present for the other four cases (with guide vane) mentioned in table1 though the Reynolds number is same for all the cases.

2. For case 1 i.e. without guide vane the fully developed flow may not exist up to the value of X/D = 3, whereas, fully developed flow exists for the value X/D = 2 corresponding to case 2, 3 and 4 (case 2, 3 and 4 conditions mentioned in the Tab. 1).

3. For case 5 (case 5 conditions mentioned in the table) also fully developed flow does not exist for its value X/D = 2.

References

[1] **Ono, A., Kimura, N., Kamide, H. and Tobita, A.**: Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition, *Nuclear Engineering and Design,* 241, 4409–4419, **2011**.

[2] **Hüttl, T. J. and Friedrich, R.**: Direct numerical simulation of turbulent flows in curved and helically coiled pipes, *Computers & fluids,* 30, 591–605, **2001**.

[3] **Chang, T-H.**: An investigation of heat transfer characteristics of swirling flow in a 180^o circular section bend with uniform heat flux, *KSME international journal,* 17, 1520–1532, **2003**.

[4] **Hellström, F.**: Numerical computations of the unsteady flow in turbochargers, **2010**.

[5] **Yamano, H., Tanaka, M., Murakami, T., Iwamoto, Y., Yuki, K., Sago, H. and Hayakawa, S.**: Unsteady elbow pipe flow to develop a flow–induced vibration evaluation methodology for Japan Sodium-Cooled Fast Reactor, *Journal of nuclear science and technology,* 48, 677–687, **2011**.

[6] **Enayet, M. M., Gibson, M. M., Taylor, A. M. K. P. and Yianneskis, M.**: Laser–Doppler measurements of laminar and turbulent flow in a pipe bend, *International Journal of Heat and Fluid Flow,* 3, 213–219, **1982**.

[7] **Anwer, M. and So, R. M. C.**: Swirling turbulent flow through a curved pipe, *Experiments in fluids,* 14, 85–96, **1993**.

[8] **Pittard, M. T. and Blotter, J. D.**: Numerical modeling of LES based turbulent flow induced vibration, in *ASME 2003 International Mechanical Engineering Congress and Exposition*, **2003**.

[9] **Qing, M., Jinghui, Z., Yushan, L., Haijun, W. and Quan, D.**: Experimental studies of orifice–induced wall pressure fluctuations and pipe vibration, *International journal of pressure vessels and piping,* 83, 505–511, **2006**.

[10] **Zhang, H., Zhang, X., Sun, H., Chen, M., Lu, X., Wang, Y. and Liu, X.**: Pressure of Newtonian fluid flow through curved pipes and elbows, *Journal of Thermal Science,* 22, 372–376, **2013**.

[11] **Ito, H.**: Pressure losses in smooth pipe bends, *Journal of Basic Engineering,* 82, 131–140, **1960**.

[12] **Imao, S., Itoh, M. and Harada, T.**: Turbulent characteristics of the flow in an axially rotating pipe, *International Journal of Heat and Fluid Flow,* 17, 444–451, **1996**.

[13] **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, 229–239, **2014**.

[14] **Homicz, G. F.**: Computational fluid dynamic simulations of pipe elbow flow, —textitUnited States, Department of Energy, **2004**.

[15] **Rahimzadeh, H., Maghsoodi, R., Sarkardeh, H. and Tavakkol, S.**: Simulating flow over circular spillways by using different turbulence models, *Engineering Applications of Computational Fluid Mechanics,* 6,100–109, **2012**.

[16] **Dutta, P.and Nandi, N.**: Effect of Reynolds Number and Curvature Ratio on Single Phase Turbulent Flow in Pipe Bends, *Mechanics and Mechanical Engineering,* 19, 5-16, **2015**.

[17] **Goodarzi, M., Safaei, M. R., Vafai, K., Ahmadi, G., Dahari, M., Kazi, S. N. and Jomhari, N.**: Investigation of nanofluid mixed convection in a shallow cavity using a two–phase mixture model, *International Journal of Thermal Sciences,* 75, 204-220, **2014**.

[18] **Safaei, M. R., Goshayeshi, H. R., Razavi, B. S. and Goodarzi, M.**: Numerical investigation of laminar and turbulent mixed convection in a shallow water–filled enclosure by various turbulence methods, *Scientific Research and Essays,* 6, 4826–4838, **2011**.

[19] **Dutta, P., Saha, S. K., Nandi, N.**: Computational study of turbulent flow in pipe bends, *International Journal of Applied Engineering Research,* 10, **2015**.

[20] **Dutta, P., Saha, S. K., Nandi, N. and Pal, N.**: Numerical study on flow separation in 90? pipe bend under high Reynolds number by k-varepsilon modelling, *Engineering Science and Technology, an International Journal,* 19, 904–910, **2016**.

[21] **Dutta, P. and Nandi, N.**: Study on pressure drop characteristics of single phase turbulent flow in pipe bend for high Reynolds number, *ARPN J. Eng. Appl. Sci,* 10, 2221–2226, **2015**.

[22] **Tu, J., Yeoh, G. H. and Liu, C.**: Computational fluid dynamics: a practical approach, *Butterworth–Heinemann*, **2012**.

[23] **Zhang, T., Zhang, Y. O. and Ouyang, H.**: Structural vibration and fluid–borne noise induced by turbulent flow through a 90 piping elbow with/without a guide vane, *International Journal of Pressure Vessels and Piping*, 125, 66–77, **2015**.