Mechanics and Mechanical Engineering
Vol. 19, No. 1 (2015) 5–16

Effect of Reynolds Number and Curvature Ratio on Single Phase Turbulent Flow in Pipe Bends

Prasun Dutta
Nityananda Nandi
Department of Aerospace Engineering and Applied Mechanics,
Indian Institute of Engineering Science and Technology, Shibpur,
PO: Botanic Garden, Dist: Howrah, West Bengal, India – 711103
pd.iiest@gmail.com

Received (11 December 2014)
Revised (16 April 2015)
Accepted (21 April 2015)

Curved pipes are very often used in hydraulic systems facilitating compact, lightweight designs. But they can also be the cause of complex secondary flows as the curvature brings change of velocity profile, generation of vortices and production of hydraulic losses. In the present study, turbulent single phase flows through circular 90 ° curved bend for different curvature ratio ($Rc/D = 1$ to 5), defined as the bend mean curvature radius (R_c) to pipe diameter (D) is investigated numerically for different Reynolds number (R_e) ranging from 1×10^5 to 10×10^5. The purpose of this study is to simulate numerically the flow pattern and characterize the swirling secondary flow in 90 ° bends. Flow simulation using CFD techniques are performed to understand these phenomena. The $k-\varepsilon$ model with SIMPLE method is used for present study. After validation of present model with published experimental data, a detail study has been performed to characterize the flow separation and the dependency of swirl intensity on Reynolds number and curvature ratio in 90 ° pipe bend for single phase turbulent flow.

Keywords: CFD, Curvature & Reynolds number effect, 90 ° pipe bend, Swril intensity, turbulent flow, velocity distributions.

1. Introduction

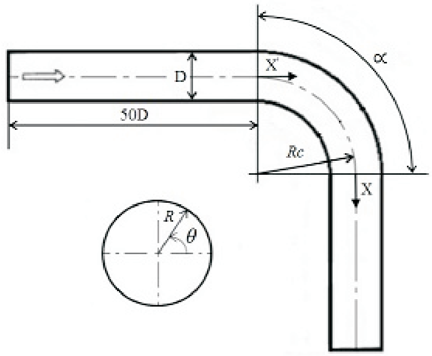
Fluid flows in a pipe bend with curvature is very common in engineering applications. They can be found in most of the e.g. fittings, HVAC appliances, turbine and blade passages and also for cooling and heating applications. It is well known that, when fluid flows in a pipe bend a secondary motion of flow is developed due to the presence of centrifugal force which leads to the formation of secondary flow and as a result the fluid particles near the surface are driven outward. This secondary flow is superimposed on its primary axial flow leading to high velocity at the outer core of the pipe bend. As it known from the literature, this secondary flow is developed due to a lack of proportion between the centrifugal force and the pressure gradient near the wall of pipe band at radial direction of flow. In general, the secondary flow in pipe bends are influenced by different parameters such as the curvature ratio (R_c/D), defined as the pipe diameter (D) over bend radius of curvature (R_c), Reynolds number (R_e), inlet flow distributions condition of the entrance flow, ie; laminar or turbulent [1]. This secondary flow results a very high frictional loss in pipe bends than in straight pipes under similar conditions. For a bend having high curvature ratio ($R_c/D > 1.5$), the velocity profile of primary axial flow is distorted and shifted to outer core of the bend from center of the bend generating a pair of counter rotating vortices which is well known as Dean vortices. For bends having small curvature ratio ($R_c/D < 1.5$), the axial flow becomes unsteady and separated due to the formation secondary flow and the flow separation occurs at the downstream of the pipe bend [2–4]. Therefore study of the characteristics of flow pattern in a pipe bend is of great significance in pipe bend design.

A number of experimental and computational study have been carried out in past on curved ducts with rectangular cross–sections, [5–7] but much less effort was for bends with circular cross–sections, which are concerned with problems in experimental work [8-9]. The applications of water flows through pipe bends are found in many engineering applications [10–11] bears testimony to this fact. Weskey [12] studied experimentally the velocity profiles at the outlet of duct with different shapes for specially design of an aircraft duct. Many researchers [9, 13–14] investigated turbulent flows in a pipe bends by means of theoretical, experiments and numerical methods. Recently in the nuclear sector due to the fatigue by the unsteady motion of the vortices, this has also attracted the interest of the researchers and they obtained very useful data in this sector [15-16]. The influence of curvature ratio (R_c/D) on the flow was studied by Al-Rafai et al. [17] using circular pipe bend of two different curvature ratio ($R_c/D = 3.49\&6.975$) for same Reynolds number ($R_e = 34132$) by means of both experimental and numerical technics. Using large eddy simulation (LES), Tanaka et al. [18] studied the flow characteristics in elbows with different curvature ratio at numerous Reynolds number. In addition a more recent survey conducted on aspect of swirl flow characteristics [19–20]. In order to perform numerical simulation of fluid flow in curved pipes, on the other hand, the Navier–Stokes equation have to be expressed in curvilinear or body fitted coordinate system as expressed by [21–22]. They provides a very useful data base for direct numerical simulation (DNS) and large eddy simulation (LES) on pipe bend. Very recent study on characteristics of secondary flow and decay of swirl intensity on pipe bends [3] and another experimental study of pressure fluctuation [23] and numerical study on pressure drop characteristics and Reynolds number dependency on pipe bend [24] provides very useful data base. From the literature study. It is observed that the investigations of secondary motion of turbulent flow in pipe bends is experimentally difficult and expensive, and alternative is to study them numerically. With the help of powerful super computers over last few years, it is now possible to the simulations of three–dimensional computations for such cases.

From the literature study it has been observed that, most of the studies concentrated only on the flow characteristics in the bend or downstream of bend for a restricted curvature ratio and Reynolds number. For accurate evaluations of flow structure and flow characteristics in a pipe bend, it is important to study with a wide range of curvature ratio and Reynolds number. The objective of present study is to simulate of the flow pattern in 90° pipe bends with different curvature ratio with different Reynolds number. The details of flow pattern throughout the bend were studied. The variations of velocity profiles with curvature ratio and Reynolds number were studied. The dependency of swirl intensity on a wide range of curvature ratio and Reynolds number are unique features of this paper.

2. Geometry and flow parameters

For the present study, five different pipe bends are considered having same inner diameter of 0.01 m but different curvature ratio ($R_c/D = 1$ to 5). The inlet & outlet length of straight pipe in the calculations was set up 50D & 20D respectively to save computational time. The velocity distributions throughout the bend and swirl intensity at the bend outlet were determined for different Reynolds number ranging from 1×10^5 to 10×10^5. The fluid medium was air having density (ρ) of 1.2647 kg/m^3 and dynamic viscosity (μ) of 1.983× 10–5 kg/m-s for validation purpose and water having density (ρ) of 998.3 kg/m^3 and dynamic viscosity (μ) of 0.001003 kg/m-s for present study with working temperature of 300 K both cases. Fully structured 3D mesh was used containing hexahedral elements, which was optimized via a grid-independence study. The value of non–dimensional distance from wall (Y^+) is controlled using a wall treatment function ($Y^+ < 3$ for a near wall cell used for present study). The bend geometry and mesh are shown in Fig. 1.



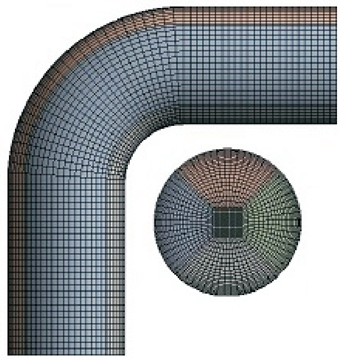


Figure 1 Schematic diagram of the bend geometry and present model with computational grid

3. Governing Equations and Numerical Methodology

3.1. *Governing Equations*

Three dimensional steady state Reynolds averaged Navier Stokes (RANS) Equations were solved using the finite volume solver. The right choice of a turbulence model is a critical when an industrial turbulent flow problem is faced, especially when this problem involves three dimensional flow phenomena, which need an accurate modelling. The second order scheme was used for the RANS Equation calculations, with a pressure velocity coupling achieved using SIMPLE algorithm. The default under relaxation factors were used to aid convergence for all models.

The governing Equations for incompressible fluid flow with constant properties are

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \tag{1}$$

$$\rho(\frac{\partial u}{\partial t} + u \cdot \nabla u) = -\nabla p + \mu \nabla^2 u + f \tag{2}$$

Eq. (1) and (2) are conservation of mass and momentum, respectively.

3.2. *Turbulence Model*

Turbulent flows are characterized by fluctuating velocity fields. These Fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. 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 attempt for the specific problem by other researchers, the accuracy needed, the available computational resources, and time.

The $k - \varepsilon$ turbulence model is adopted for present study as $k - \varepsilon$ turbulence model performs better for single phase flow in pipe bend [3, 25–26]. 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 $k - \varepsilon$

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

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

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

The Eq. (3) and (4) also consist of some adjustable constants, these are as follows

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

3.3. *Boundary Conditions*

Three types of boundary conditions has been specified for the computational domain. At the inlet, the measured inlet velocity (U_{in}), the turbulent kinetic energy, $k_{in} = 1.5(I{*}U_{in})^2$ where I is the turbulence intensity, and the Specific Dissipation Rate $\varepsilon_{in} = (C_\mu k^{3/2})/0.3D$ are given. The turbulence intensity has been calculated using the following formula: $I = 0.16(R_e)^{-0.125}$. At the wall boundaries, the non-slip conditions has been applied. At the outlet, the gradient of flow variables in the flow direction has been considered as zero.

4. Results and Discussions

The main objective of the present study is to characterize the effect of Reynolds number and curvature ratio on single phase turbulent flow in a 90 ° pipe bend through numerical simulation. The results of the mean velocity and swirl intensity along the different positions of the bend in central symmetry plane are presented in this section.

4.1. *Validation*

At the very beginning of our study, our present model and simulation setup are first validated against the existing experimental and numerical data by [9, 3, 18]. For that intension, same geometrical configuration of [9] is adopted. In their experiment, they used a circular cross sectioned 90 ° bend with a curvature ratio (R_c/D) of 2 and the measurements of velocities were performed at a Reynolds number of 60000. For the validation of our present model, the simulation is performed on a computational mesh containing total 2.85 million hexahedral elements which was optimized via a grid-independence study. The value of non-dimensional distance from wall (Y^+) is strictly controlled using a wall treatment function ($Y^+ <3$ for a near wall cell used for present study). The mean axial velocity profile normalized with inlet velocity (Fig. 1) show very good agreement with both experimental data of [2] and numerical data of [3, 18]. From the validation part it has been seen that present model is in close approximation with the published results and our methodology to perform present simulation is correctly achieved, hence this model and methodology has been used for further analysis.

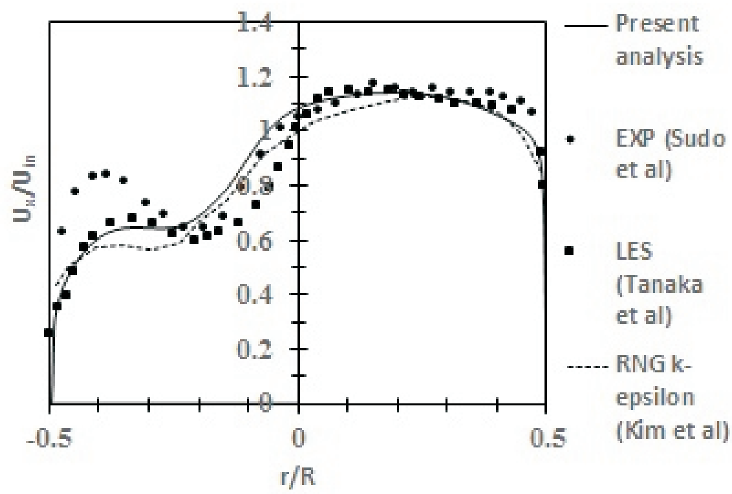


Figure 2 Comparison of present model with experimental and numerical results

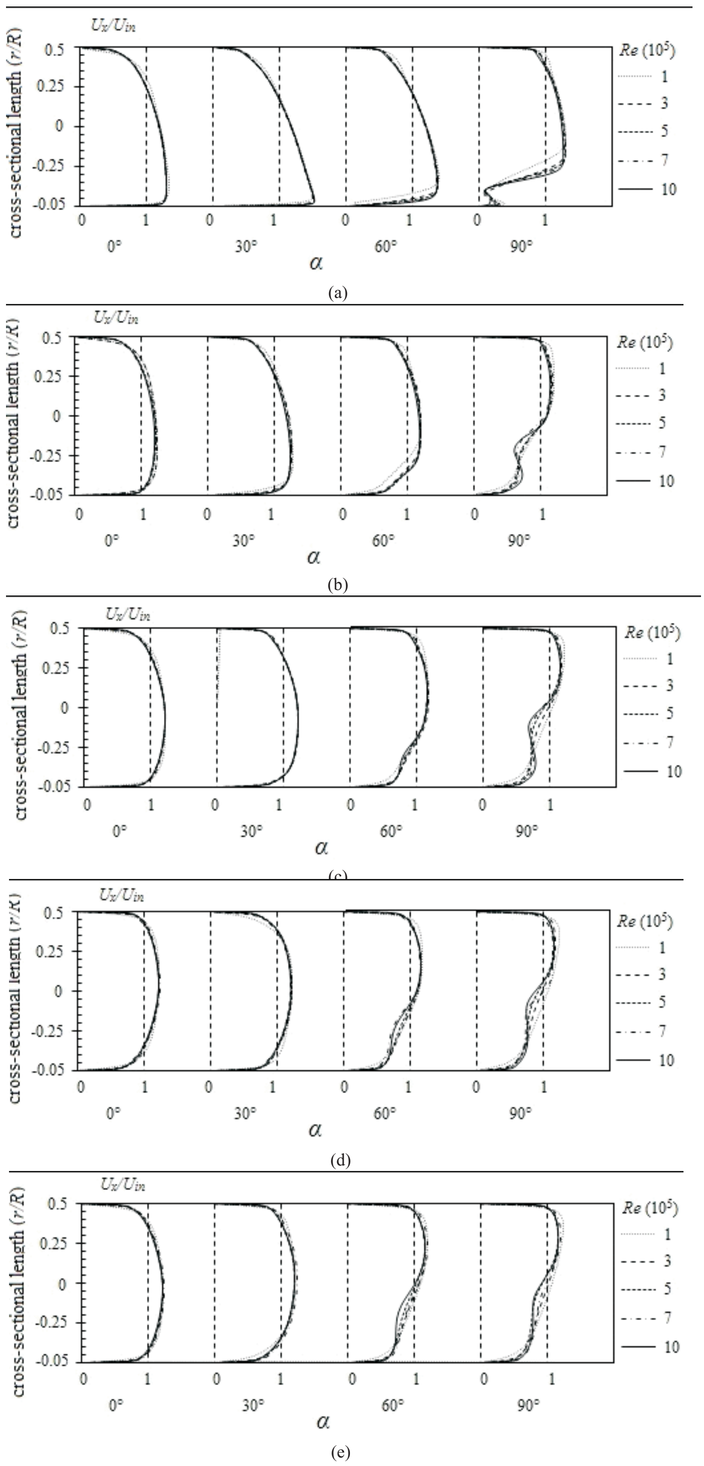


Figure 3 Normalized velocity profiles at different positions throughout the bend for different R_e & R_c/D: a) $R_c/D = 1$, b) $R_c/D = 2$, c) $R_c/D = 3$, d) $R_c/D = 4$, e) $R_c/D = 5$

4.2. *Study on mean velocity*

Fig. 3 (a–e) illustrate normalized mean velocity profiles of turbulent flow into the 90° pipe bend for different Reynolds number and curvature ratios. The negative value of y–axis indicates the inner core of the bend and the mean velocity is normalized with inlet velocity. At the inlet of the bend ($\alpha = 0^\circ$), fluid is accelerated near the inner core and velocity acceleration depicted at the outer core of the bend as expected due to the longitudinal pressure gradient for bends with low curvature ratio ($Rc/D = 1$ & 2). Reynolds number is found as a weak function for bends with low curvature ratio. After $\alpha = 60^\circ$ bends with high curvature ratio ($R_c/D = 3$ to 5), the mean velocity is shifted upward due to the faster moving fluid near the outer core as expressed by [9]. Between 75° and 90° plane, the flow has detached from the inner wall, and the separated flow region grows as the fluid moves downstream, causing flow acceleration along the outer wall. The main characteristic of this flow in this zone is the appearance of the camel back shapes in the velocity distribution. These shapes are the result of loss of the stream wise momentum due to the secondary flow forming a strong downward motion from the symmetry plane to the bottom (or top) wall. It is found that, flow tends to recover its fully developed shape while the Reynolds number is increasing. The normalized velocity at symmetry plane of the bend decreases with increasing Reynolds number. This observation is also consistent with recent experiments [27, 3] and it is also true for a high range of Reynolds number and curvature ratio. Hence, it can be concluded that for higher values of Reynolds number, pipe curvature effects are reducing.

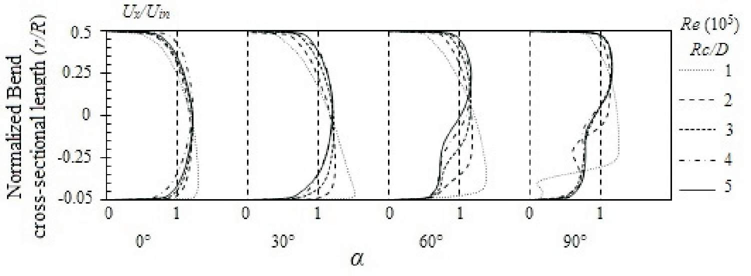


Figure 4 Normalized velocity profiles at different positions throughout the bend for different curvature ratio

To study the effect of curvature ratio on velocity profile, Fig. 4 shows the normalized mean velocity profile with inlet velocity for single Reynolds number ($R_e = 1{\times}10^5$) with all five bends ($R_c/D = 1$ to 5). At the outlet of bend ($\alpha = 90^\circ$), reverse flow has been 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. It is found that the acceleration of the velocity is higher for bends having low curvature ratio ($R_c/D = 1$). At the downstream of the bend, tendency of acceleration of velocity at the inner core, to recover its fully developed shape is high for bends with high curvature ratio

($R_c/D = 5$). Hence, it can be concluded that the ability to overcome the unsteady and complex flow pattern is more for bends with high curvature ratio.

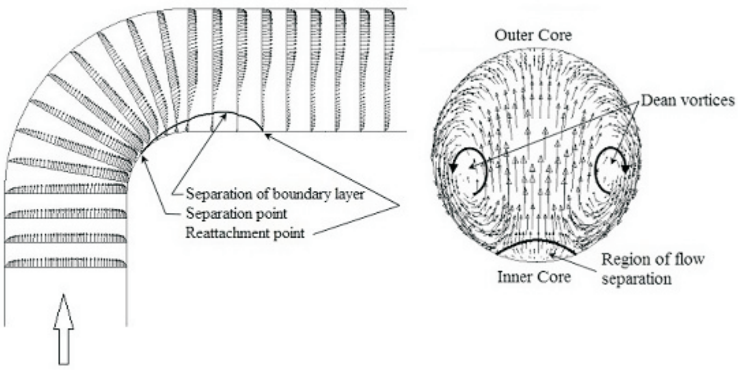


Figure 5 Normalized flow velocity vector fields

Fig. 5 shows the normalized flow velocity vector fields at the central symmetry plane and at the bend outlet ($\alpha = 90^\circ$) for $Re = 1{\times}10^5$, $R_c/D = 1$. From $\alpha = 60^\circ$ to $\alpha = 90^\circ$, the mean velocity is shifted upward due to the faster moving fluid near the outer core as expressed by [9]. Secondary flow pattern can clearly be observed in this section. At the inner core of the bend, a low velocity region was observed, where the flow pattern was extremely unsteady and complex. This region is concluded to be the separated region [23] and the size of this region is more for bends with low curvature ration than others. It means that the tendency of flow separation is more for bends having low curvature ratio at the inner core of the bend. Due to the relation between the centrifugal force with high velocity and pressure gradient on the flow, two identical counter rerating Dean vortices were also found.

4.3. *Study on mean velocity*

It is already known that the secondary flow consists two identical counter-rotating vortices or swirl. In general, the strength of a swirl on a cut plane normal to the axil flow direction is represented by swirl number [28]. The swirl number is the area–averaged flux of angular momentum and it is used to quantify the strength of a swirl around the axis of main flow [9].

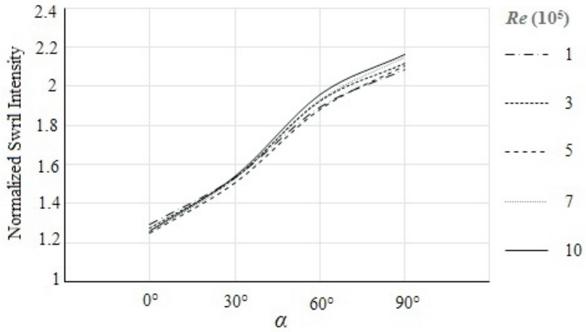


Figure 6 Normalized swirl intensity throughout the bend (Rc/D)

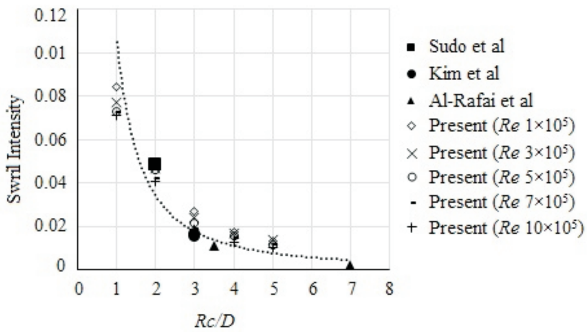


Figure 7 Influence of curvature ratio on swirl intensity at bend outlet

To define swirl intensity, [9] used the magnitude of velocity tangential to the pipe section and eq. (5) was used by [3] which was also used for present study.

$$I_s = \frac{\int \left[\vec{U} - (\vec{U} \cdot \hat{n})\hat{n}\right]^2 dA}{U_b^2 \int dA} \tag{5}$$

where I_s is the swirl intensity, is the vector of the flow velocity and is a unit vector normal to the pipe section area.

To evaluate the dependency of swirl intensity on Reynolds number in a pipe bend, the intensity of secondary flow calculated from equation (5) at different plane through the bend is normalized by the value of intensity at reference plane ($x'/D = -17.6$). The normalized swirl intensity for bend with $R_c/D = 1$ is shown in Fig. (6). It can be observed that the swirl intensity is very low dependence on Reynolds number.

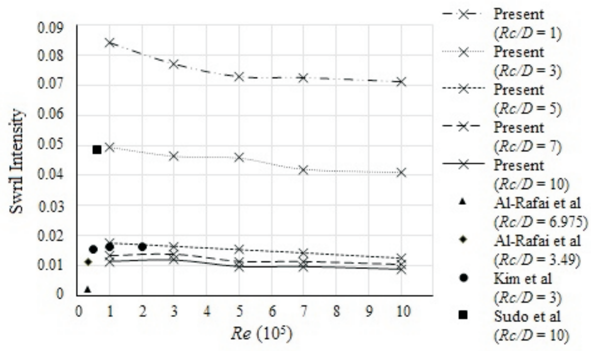


Figure 8 Influence of Reynolds number on swirl intensity

To study the effect of Reynolds number & curvature ratio on swirl intensity at the exit of the bend ($\alpha = 90^\circ$), the results are compared with previous experimental results of [17, 9, 3]. Additionally present study results are also plotted in Fig. (7) and Fig. (8). It can be found that swirl intensity has a weak dependence on Reynolds number and a high dependence on curvature ratio as expressed by [3] and it is true for a higher range of Reynolds number and curvature ratio also.

5. Conclusions

In the present study turbulent flow of single phase incompressible fluid through 90° pipe bend has been simulated numerically using $k - \varepsilon$ turbulent model. The validation of 3D models used for present study with experiments and numerical results reported by [9] and [3, 18] indicates a good agreement. From the present study, it is found that the normalized mean velocity profile has a low dependency on Reynolds number for low curvature ratio and for high curvature ratio it tends to recover its fully developed shape while Reynolds number is increasing. Hence, it can be concluded that, for higher values of Reynolds number, bend curvature effects are reduce. The tendency of flow separation is more for bends having low curvature ratio and the ability to overcome the unsteady and complex flow pattern is more for bends with high curvature ratio. The swirl intensity at the exit of the bend is found as a strong function of the bend curvature ratio and a weak function of Reynolds number for present study range. However additional studies are required to provide a correlations between swirl intensity and bend curvature ratio.

References

[1] **Lai, Y. G., So, R. M. C. and Zhang, H. S.**: Turbulence–driven secondary flows in a curved pipe, *Theoretical and Computational Fluid Dynamics*, 3, 163–180, **1991**.

[2] **Hellström, L. H., Sinha, A. and Smits, A. J.**: Visualizing the very–large–scale motions in turbulent pipe flow, *Physics of Fluids (1994-present)*, 23(1), 011703, **2011**.

[3] **Kim, J., Yadav, M. and Kim, S.**: Characteristics of Secondary Flow Induced by 90 $^\circ$ Elbow in Turbulent Pipe Flow, *Engineering Applications of Computational Fluid Mechanics*, 8(2), 229–239, **2014**.

[4] **Dutta, P., Saha. K. S.and Nandi, N.**: Numerical study of curvature effect on turbulent flow in 90 $^\circ$ pipe bend, *Proceedings of Sixth International Conference on Theoretical, Applied, Computational and Experimental Mechanics*, IIT Kharagpur, India, 44–45, **2014**.

[5] **Iacovides, H., Launder, B. E. and Li, H. Y.**: The computation of flow development through stationary and rotating U-ducts of strong curvature, *International Journal of Heat and Fluid Flow*, 17(1), 22–33, **1996**.

[6] **Raisee, M., Alemi, H. and Iacovides, H.**: Prediction of developing turbulent flow in 90 $^\circ$-curved ducts using linear and non-linear low-Re k–ε models, *International journal for numerical methods in fluids*, 51(12), 1379–1405, **2006**.

[7] **P. Rudolf,**: Modeling of Secondary and Separated Flow in Curved Channels of Rectangular Cross–Section, *Proceedings of Topical Problems of Fluid Mechanics*, Prague, **2004**.

[8] **Sudo, K., Sumida, M. and Hibara, H.**: Experimental investigation on turbulent flow through a circular–sectioned 180 bend, *Experiments in Fluids*, 28(1), 51–57, **2000**.

[9] **Sudo, K., Sumida, M. and Hibara, H.**: Experimental investigation on turbulent flow in a circular–sectioned 90 $^\circ$ bend, *Experiments in Fluids*, 25(1), 42–49, **1998**.

[10] **Naphon, P. and Wongwises, S.**: A review of flow and heat transfer characteristics in curved tubes. *Renewable and sustainable energy reviews*, 10(5), 463–490, **2006**.

[11] **Crawford, N., Spence, S., Simpson, A. and Cunningham, G.**: A numerical investigation of the flow structures and losses for turbulent flow in 90 $^\circ$ elbow bends, *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 223(1), 27–44, **2009**.

[12] **Weske, J. R.**: *Experimental Investigation of Velocity Distributions Downstream of Single Dust Bends* (No. NACA-TN-1471). NATIONAL AERONAUTICS AND SPACE ADMINISTRATION WASHINGTON DC., **1948**.

[13] **Sakakibara, J. and Machida, N.**: Measurement of turbulent flow upstream and downstream of a circular pipe bend, *Physics of Fluids (1994-present)*, 24(4), 041702, **2012**.

[14] **Kalpakli, A., Örlü, R. and Alfredsson, P. H.**: Dean vortices in turbulent flows: rocking or rolling? *Journal of visualization*, 15(1), 37–38, **2012**.

[15] **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(11), 4409–4419, **2011**.

[16] **Yuki, K., Hasegawa, S., Sato, T., Hashizume, H., Aizawa, K. and Yamano, H.**: Matched refractive-index PIV visualization of complex flow structure in a three-dimentionally connected dual elbow, *Nuclear Engineering and Design*,241(11), 4544–4550, **2011**.

[17] **Al–Rafai, W. N., Tridimas, Y. D.and Woolley, N. H.**: A study of turbulent flows in pipe bends, *Proceedings of the Institution of Mechanical Engineers. Part C. Mechanical engineering science*, 204(6), 399–408, **1990**.

[18] **Tanaka, M. A., Ohshima, H.and Monji, H.**: Numerical Investigation of flow structure in pipe elbow with large eddy simulation approach, *ASME 2009 Pressure Vessels and Piping Conference*, pp. 449–458, American Society of Mechanical Engineers, **2009**.

[19] **Chang, T. H. and Lee, H. S.**: An experimental study on swirling flow in a 90 degree circular tube by using particle image velocimetry, *Journal of visualization*, 6(4), 343–352, **2003**.

[20] **Pruvost, J., Legrand, J. and Legentilhomme, P.**: Numerical investigation of bend and torus flows, part I: effect of swirl motion on flow structure in U–bend, *Chemical engineering science*, 59(16), 3345-3357, **2004**.

[21] **Hüttl, T. J. and Friedrich, R.**: Direct numerical simulation of turbulent flows in curved and helically coiled pipes, *Computers & fluids*, 30(5), 591–605, **2001**.

[22] **Noorani, A., El Khoury, G. K. and Schlatter, P.**: Evolution of turbulence characteristics from straight to curved pipes, *International Journal of Heat and Fluid Flow*, 41, 16–26, **2013**.

[23] **Shiraishi, T., Watakabe, H., Sago, H. and Yamano, H.**: Pressure fluctuation characteristics of the short-radius elbow pipe for FBR in the postcritical Reynolds regime, *Journal of Fluid Science and Technology*, 4(2), 430–441, **2009**.

[24] **Dutta, P., Banerjee, S., Santra, A. and Nandi, N.**: Numerical Study on Pressure Drop Characteristics of Turbulent Flow in Pipe Bend, *Proceedings of Aspects of Mechanical Engineering and Technology for Industry*, NERIST, Arunachal Pradesh, 1, 381–386, **2014**.

[25] **Homicz, G. F.**: Computational Fluid Dynamic Simulations of Pipe Elbow Flow, United States. Department of Energy, **2004**.

[26] **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(1), 100–109, **2012**.

[27] **Zagarola, M. V. and Smits, A. J.**: Scaling of the mean velocity profile for turbulent pipe flow, *Physical review letters*, 78(2), 239, **1997**.

[28] **Chigier, N. A. and Beer, J. M.**: Velocity and static–pressure distributions in swirling air jets issuing from annular and divergent nozzles, *Journal of Fluids Engineering*,86(4), 788-796, **1964**.