Journal of Applied Fluid Mechanics, Vol. 14, No. 4, pp. 1995-1208, 2021.
Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
https://doi.org/10.47176/jafm.14.04.32148

Effect of Guide Vane on Turbulence Characteristics for Single-Phase Flow through a 90-Degree Pipe Bend

S. Kumar Saha† and N. Nandi

Department of Aerospace Engineering & Applied Mechanics, Indian Institute of Engineering Science and Technology, Shibpur, Howrah-711103, India

†*Corresponding Author Email: sks.iiest@gmail.com*

(Received September 1, 2020; accepted February 26, 2021)

ABSTRACT

The present study expresses the turbulent flow characteristics through a 90° pipe bend using a numerical method by determining the solutions for Reynolds Averaged Navier-Stokes (RANS) expression using the k-ω (SST) turbulence model. For that purpose, numerical analysis has been carried out by solving RANS equations using ANSYS FLUENT 16.2, considering incompressible fluid in turbulent flow conditions. Simulations have been carried out for three different Reynolds number ranging from 1×10^5 to 10×10^5 at three different bend curvature ratios (Rc/D = 1, 1.5, and 2). Pipe bends with guide vane are generally used where flow separation and space problem makes an issue in mechanical design. The presence of guide vane inside the bend positively suppressed the flow separation and presence of cross-flow which can cause the engine to run off design, thus reducing the engine efficiency. So, to observe the effect of guide vane and its position on turbulence characteristics, four different positions of guide vane inside the bend are considered in the present study. At first, an analysis was led to make sure that the results obtained from the present numerical model are reliable and in line with previous results obtained from similar published experiments and numerical work. Research has been conducted to find out the impact of Reynolds number, bend curvature ratio and position of guide vane on different turbulence characteristics namely; turbulent kinetic energy, turbulent intensity, and wall shear stress at bend outlet position. In general, the turbulent intensity is found larger for the lower bend curvature ratio at the inner wall curvature side. Results for turbulent kinetic energy have similarities in results with turbulent intensity. Significantly, the wall shear stress represented a strong dependency on the circumferential angle at the bend outlet cross-section, and curvature ratio rather than Reynolds number and guide vane positions.

Keywords: 90° pipe bend; Bend curvature ratio; Guide vane; k-omega (SST) turbulence model; Numerical analysis; Turbulence characteristics.

NOMENCLATURE

D	diameter	β_1, β_2	turbulence model constant
F_1, F_2	blending function in SST turbulence model	ω	specific rate of turbulent dissipation
I	turbulent intensity	ρ	fluid density
Rc	radius of bend curvature	μ	molecular viscosity
Re	Reynolds number	σ	diffusion coefficient
U	mean velocity	$\sigma_{\omega 1}$, $\sigma_{\omega 2}$	turbulence model constant
P_k	production term for turbulent kinetic energy	μ_t	turbulent eddy viscosity
S	shear strain rate	σ_{k1}, σ_{k2}	turbulence model constant
ε	turbulent dissipation	Y^+	normalized wall distance
k	turbulent kinetic energy		

1. INTRODUCTION

Pipelines are generally used in many industrial processes to convey supplies from one place to another. 90-degree pipe bends are frequently used in the piping network to change the direction of flow. It also takes an important role in the design of a nuclear reactor cooling system, space shuttle engine, etc. So, for any pipeline system, one of the most important parts is the pipe bends as they deliver lightness in the fluid transmission. Hence a comprehensive understanding of fluid dynamics in pipe bends is essential for good pipe design and pipeline arrangements. Studies on turbulent flow through pipe bends for different geometrical changes are vital in determining and refining the performance of the piping network and reducing the energy losses during flow. Flow in bent conduits shows that a secondary motion is produced by the lack of equilibrium between the inertial forces and the developed adverse pressure gradient. The reason for the observation is that, during flow, the bend is solely the source of a secondary flow, illustratively, the bend causes a transversal transport of stream-wise momentum, and the increase in momentum is the reason for the increased friction in the pipe. On the contrary, during turbulent flow, the bend-induced vortices add themselves to the pre-existing turbulence present in the flow.

The purpose of the present work is to conduct a numerical study about the changes in turbulent characteristics for single-phase turbulent flow through a 90-degree pipe bend with a guide vane at different position inside the bend. A detailed review of the literature has been carried out to get an idea regarding the present study. From the previous literature, it has been observed that the measurement techniques employed mainly include Laser Doppler Velocimetry (LDV) and hot-wire anemometry, whereas wall shear gauges, pressure taps, and Pitot tubes have also been used for experimental study. At the same time, turbulence models have also been utilized as the experiments, mainly the most common ones (standard k-ε or k-ω). From many previous experimental studies (Enayet *et al.* 1982; Azzola *et al.* 1986; Lee *et al.* 2007), it was observed for 90-degree pipe bend cases that at 90-degree position, the maximum of the velocity was found near to the outer wall and remains so along with the downstream. The mean velocity profile becomes more uniform and the strength of the secondary flow decreases (Sudo *et al.* 1998; Anwer *et al.* 1989; Hellstrom *et al.* 2011) as the flow moves towards downstream means the distance from the bend outlet increases. It should also be noted that mean velocity, turbulence intensity, and Reynolds stresses were found to be similar up to a bend angle of 60-degree (Sudo *et al.* 2000) for both 90° and 180° bends. The most extensive studies on turbulent statistics of flows in 90-degree and 180-degree bends are the ones by Sudo *et al.* (1998, 2000).

An attempt to predict the flow through a bend by applying turbulence modeling was first made by Patankar *et al.* (1975) using the k-ε turbulence model. In the study, Patankar *et al.* (1975) compared the numerical results with experiments given by Rowe (1970) for a developing flow in a 180-degree bend. Al-Rafai *et al.* (1990), did both the experimental and numerical calculations for a turbulent flow at Re = 3.4×10^4 in two bends. Measurements were performed employing LDV for the experiment, whereas the k-ε model was employed for calculating the flow numerically. The main results involved mean and r.m.s. axial velocities which showed the predictable result that the secondary flow is stronger in the bend with comparatively the higher curvature. The performance of various turbulence models was also tested later by many researchers (Pruvost *et al.* 2004; Hellström and Fuchs 2007; Sugiyama and Hitomi 2005; Pellegrini *et al.* 2011; Röhrig *et al.* 2015). Several researchers have done studies about turbulent flows in curved sections of pipes by the use of theoretical, experimental, and numerical models, and recently a very convenient database for Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) on pipe bends are provided by these studies (Hüttl and Friedrich 2001; Noorani *et al.* 2013; Di Piazza and Ciofalo 2011; Di Liberto *et al.* 2013).

Researchers have already done a few investigations about the effect of guide vane on different flow parameters. The flow in a two-dimensional 90° circular-arc bend with guide vane was studied by Kotb *et al.* (1988) numerically. In their study, they have considered both laminar and turbulent flows to observe the size and occurrence of the separation regions. This study shows that the emerging velocity distribution is more uniform and the presence of a guide vane is shown to mitigate the formation of regions of flow separation. Liou *et al.* (2001) studied the effects of Guide-Vane number for flow in a three-dimensional 60° curved side-dump combustor inlet and the Reynolds number was 2.53×10^4. The guide-vane number and Reynolds number were the main parameters examined. The above-mentioned study reveals that three guide vanes should be installed to eliminate flow separation in the curved combustor inlet. Moujaes *et al.* (2009) studied the effects on pressure drop for turning vanes in 90° duct elbows utilizing both CFD predictions and experimental comparisons. A study on performance analysis due to the insertion of guide vanes inside a rectangular elbow cross-section was carried out by Nugroho *et al.* (2016). Structural vibration and fluid-borne noise-induced for flow through a 90° piping elbow with/without a guide vane in turbulent flow condition was studied by Zhang *et al.* (2015) using the LES model. The aforementioned numerical study showed that the right location of the guide vane was effective in reducing vibration and flow-induced noise in the 90° piping elbow with water. A recent study regarding the effect on in bend pressure drop and post bend heat-transfer for a partial blockage at the bend inlet has been investigated by Gorman *et al.* (2018) by employing SST k-ω turbulence model. Valsala *et al.* (2019) conducted a study on the reduction in pressure losses in pipe bends using one and multiple guide vane conditions. Their work focuses on flow characteristics on a 90° curved bend by solving the RANS equation using turbulence treatment with the k-ω SST (Shear Stress Transport) model. Turbulent flow characteristics of single-phase, incompressible flows in a 90° circular pipe bend with guide vane at different positions are not fully clarified yet. In this study, an attempt has been made to study the effect of guide vane and its positions on turbulent parameters. After getting a brief idea of previous research works and the necessary theoretical background present study has been carried out.

Changes in flow structure on the flow field are presented and discussed in the form of graphical and plot representation in the results and discussion section. Finally based on the present results, concluding remarks are drawn from the study followed by the conclusion section.

2. PROBLEM DESCRIPTION

The present problem is focused on the effect of Reynolds number, curvature ratio, and position of guide vane on turbulent parameters for single-phase, steady incompressible internal turbulent flow through a pipe bend. A representative diagram of the physical situation in computational domain is shown in Fig. 1.

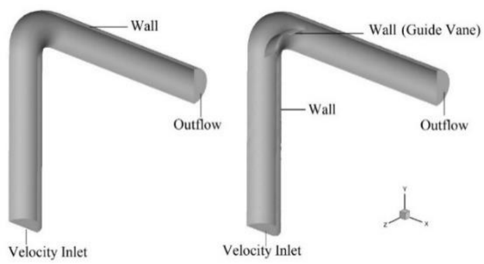


Fig. 1. A representative diagram of the bend geometry (a) Without Guide Vane and (b) With guide vane

Figure 1 reveals that the present system consists of three (without guide vane case) to four (with guide vane case) individual entities: (a) a straight circular section at the upstream to deliver fluid flow to the bend inlet, (b) the bend section, (c) guide vane that is situated at the bend portion starting from bend inlet to bend outlet and again (d) a straight circular section to receives the outflow from the bend outlet. Here 90° pipe bend with a diameter (D) 0.1 m and three different curvature ratios (1, 1.5 and 2) has been considered as the geometry for the present work. Since the pipe bends or elbows having curvature ratio values greater than 1.5 are known as long-radius pipe elbows, hence the 90° pipe bends considered here fall respectively into the short-radius, medium-radius and long-radius categories. Moreover, the most used 90-degree pipe bends in practical fields are in this curvature ratio range. So primarily only three different curvature ratios have been considered in the present study. Straight pipe length equal to 20 times of pipe hydraulic diameter at the upstream and 10 times at downstream of the 90° pipe bend has been considered. Total four different guide vane positions have been considered for the present study varying three different Reynolds number. The Reynolds number will be defined as

$$Re = \rho DU / \mu \tag{1}$$

In which ρ and μ are the density and viscosity of the flowing fluid, respectively and U is the mean velocity of the flow. The positions of the guide vanes are determined according to the radius ratio (ratio between nominal elbow radius to the radius of elbow curvature) suggested in established literature (Zhang *et al.* 2015). Different radius ratio values, as well as the positions of guide vane from the virtual centre of the bend curvature, are given in table 1. So, twelve different geometry models of pipe bend with guide vane at different positions and three simple bend geometry i.e. without guide vane have been modeled in the computational domain. The working fluid for the present study has been considered as a single-phase fluid *i.e.* water at 20° C having dynamic viscosity (μ) 0.001003 kg/m-s and density (ρ) 998.2 kg/m^3. Table 1 shows the cases with the different geometrical configurations according to the guide vane and its position inside the bend. Three different Reynolds number ranging from 1×10^5 to 1×10^6 have been taken into consideration and numerical simulation carried out for each case to study the effect of it. Since the present study absolutely focuses on the effect of geometrical configuration, mainly the presence and position of guide vane, so the wall of pipe bend and guide vane had considered as smooth wall. Therefore, material properties for all the numerical simulations that have carried out for the present study had kept constant throughout the investigation. Although the role of material property acts as one of the basic factor on determining flow characteristics, which was not discussed in detail in the present study as because it is beyond the scope of study.

3. METHODOLOGY AND VALIDATION

Turbulent flows are generally designated by the fluctuations of the velocity which are highly irregular. Different transport quantities like momentum, energy, etc. also fluctuate for this fluctuation. In industries, maximum fluid flow applications are highly turbulent. Some of the cases are dealt with under a high Reynolds number condition. In this flow condition, the speed of the fluid at a point undergoes a continuous change in both magnitude and direction, hard to analyze the problem directly (Spalart 2000). So, to get reasonable results, a proper turbulence model should be applied to solve the RANS equation numerically (Speziale 1998). From the previous literature (Dutta *et al.* 2016; Menter 1994; Reynolds 1987) it has been identified that generally the RANS models: k-ε, k-ω, Reynolds stress model (RSM) and Large Eddy Simulation (LES) (Rütten *et al.* 2001) are used to investigate the turbulent flow through a pipe bend. Wilcox *et al.* (1998) in their study that the results obtained using the k-ω model in the logarithmic region are superior to the results obtained by the k-ε model in equilibrium adverse pressure gradient flows. Sparrow *et al.* (2008) shows in their study that the SST k-ω turbulence model also gives reasonably good results for the flows that

Table 1 Position of Guide Vane for different Curvature ratios.

Curvature Ratio 1		Value of radius ratio	Position of guide vane (m)
Case 1		0	No guide vane
Case 2	With Guide Vane	0.175	0.068
Case 3		0.308	0.081
Case 4		0.493	0.099
Case 5		0.738	0.124
Curvature Ratio 1.5		Value of radius ratio	Position of guide vane (m)
Case 1		0	No guide vane
Case 2	With Guide Vane	0.175	0.118
Case 3		0.308	0.131
Case 4		0.493	0.149
Case 5		0.738	0.174
Curvature Ratio 2		Value of radius ratio	Position of guide vane (m)
Case 1		0	No guide vane
Case 2	With Guide Vane	0.175	0.168
Case 3		0.308	0.181
Case 4		0.493	0.199
Case 5		0.738	0.224

handling laminar regions contained within a turbulent zone. Swienty et al. (2015) established in their study that the SST (Shear Stress Transport) k-ω model showed a good correlation with the measurements for resolving shear and swirl flows in pipes. Henceforth SST k-ω turbulence model could be the proper model compare to the others to compute a flow field that exhibits separated flow, adverse pressure gradient, strong curvature, etc. So, the present study has been carried out considering the SST k-ω turbulence model.

3.1 Governing equations

Based on the substantial literature and experience, the SST k-ω turbulence model is adopted for the present numerical solution. The equations, which govern incompressible fluid flow, are the conservation of mass and momentum. The common approach to express the momentum equation is the Reynolds-Averaged Navier-Stokes equations. Equations for turbulent kinetic energy (k) and specific dissipation rate (ω) with constant properties are shown below.

Transport equation for turbulent kinetic energy

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=P_k-\beta^* k\omega+\frac{\partial}{\partial x_j}\left[\left(\nu+\sigma_k\nu_T\right)\frac{\partial k}{\partial x_j}\right] \quad (2)$$

Transport equation for the specific dissipation rate

$$\frac{\partial \omega}{\partial t}+U_j\frac{\partial \omega}{\partial x_j}=\alpha S^2-\beta\omega^2+\frac{\partial}{\partial x_j}\left[\left(\nu+\sigma_\omega\nu_T\right)\frac{\partial \omega}{\partial x_j}\right]+$$

$$2\left(1-F_1\right)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \quad (3)$$

Where F_1 is the blending function given by

$$F_1=tanh\left\{\left\{min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right),\frac{4\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \quad (4)$$

$$CD_{k\omega}=max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i},10^{-10}\right) \quad (5)$$

$$F_2=tanh\left[\left[max\left(\frac{\sqrt{k}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right)\right]^2\right] \quad (6)$$

$$P_k=min\left(\tau_{ij}\frac{\partial U_i}{\partial x_j},10\beta^* k\omega\right) \quad (7)$$

The turbulent eddy viscosity (μ_t) for the SST model is defined as follows

$$\mu_t = \frac{a_1 k}{max(a_1\omega, SF_2)} \tag{8}$$

Where S is the vorticity magnitude and F_2 is the blending function

The model constant are as follows taken from well-known literature (Menter, 1993, 1994),

$\beta^* = 0.09, \quad \beta_1 = 3/40, \quad \beta_2 = 0.0828$

$\sigma_{k1} = 0.85, \quad \sigma_{k2} = 1,$

$\sigma_{\omega 1} = 0.5, \sigma_{\omega 2} = 0.856$

3.2 Boundary conditions

To solve any fluid flow problem numerically, initial and boundary conditions for the flow variables are required to define the flow in the computational domain. Mainly three different boundary conditions have been applied for the present study. These are 'Velocity Inlet', 'Outflow', and 'Wall' boundary conditions. At the upstream inlet section of the bend geometry, velocity inlet boundary condition, i.e. a Dirichlet boundary condition (DBC) is applied without any perturbation. For that reason, flow velocity at the inlet (U) has been calculated according to the Reynolds number consideration (Re=ρDU/μ). For the velocity inlet, the velocity vectors are specified by using magnitude and flow direction. Turbulent intensity (I=0.16×Re-0.125) of the turbulence condition and the hydraulic diameter of the pipe bend (D) are also given in the velocity inlet boundary condition part. At the downstream outlet, the outflow boundary condition has been applied in the present study. No-slip boundary condition has been applied at the pipe and guide vane wall considered that the fluid will have zero velocity at the body surface or very low near to the wall.

3.3 General mesh details

A three-dimensional structured mesh with the hexagonal grid has been adopted containing a minimum of 2.5 million number of elements for all the simulations of the present study after optimizing the total no. of elements and nodes via grid independent study. The height of the first cell at the wall has a significant role in the simulation results. So the size of the first layer of grid cells has been calculated and applied using wall function the first cell height has been calculated by aiming average normalized wall distance (y+) lies below 20 for this Reynolds number range. A further study has been carried out to check mesh refinement for all the cases. The standard methodology of (Roache 1994, 1998) was used to examine the convergence of the grid system. It was found from the grid convergence test that the finest grid achieved an error smaller than 3%.

3.4 Experimental verification details

The solutions of the governing equations for the present study were carried out considering SST k-ω turbulence model by means of ANSYS FLUENT 16.2. It is quite necessary to verify the results provided by the present numerical model with data from well-executed experiments. Results obtained from the present simulation have been compared with the existing experimental (Feng *et al*. 2014 and Kim *et al.* 2014) and numerical (Feng *et al.* 2014 and Tanaka *et al.* 2009) data. Figure 2a shows the comparison of relative total pressure at the outer and inner position of the elbow obtained from the present simulation for Reynolds number 4.7×10^4 with the experimental and numerical data given by Feng *et al.* (2014). Results on velocity distribution at bend outlet have been compared with the results given by Kim *et al.* (2014) Tanaka *et al.* (2009) because the available data are reported at this location. There is little variation shown between the experimental and numerical results in Fig. 2. These differences may come due to the reasons mentioned below. First of all, it is quite possible to maintain constantly the same environmental condition (temperature of surrounding as well as working fluid) throughout the experiments. The working fluid *i.e.* water considered in the present numerical study is chemically pure water. These could be the causes of the discrepancies between the experimental and numerical results. From Fig. 2 (a-b), it is clear that the results obtained from the present model are in close approximation with the published result. Hence further study can be carried out considering the same mesh configuration and methodology

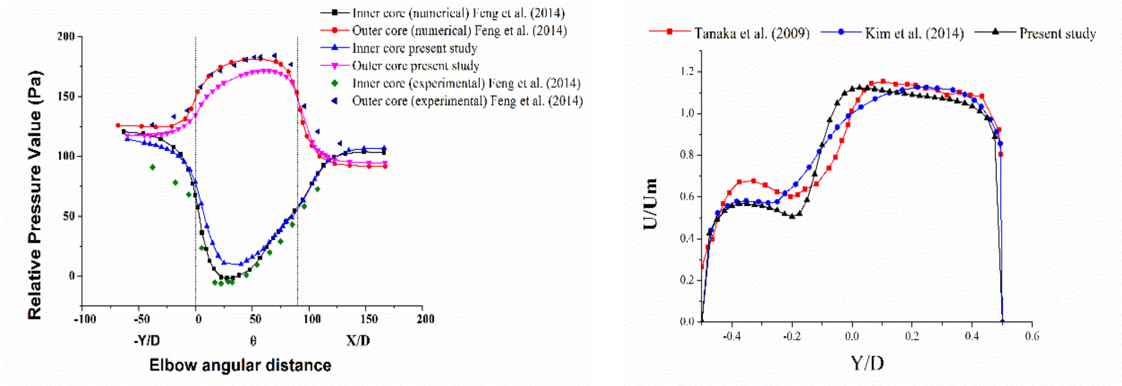
4. Results and Discussion

The purpose of the present work was to study the effect on turbulence characteristics for flow-through 90° pipe bends considering different flow conditions and geometrical configurations. In the present study, three different turbulence parameters have been taken into consideration, these are (i) Turbulent Intensity, (ii) Turbulent Kinetic Energy (TKE), and (iii) Wall shear stress. Three different Reynolds number, as well as curvature ratios with four different geometrical position of guide vane inside the bend, have been considered as mentioned earlier. Results regarding turbulent characteristics are presented and discussed in three different segments.

At first, the results are presented for the effect of Reynolds number on three different turbulent parameters and then followed by the effect of curvature ratios and positions of guide vane. All the results obtained from the study are presented in non-dimensional form. Turbulent intensity is normalized by the calculated value of intensity using the equation $I = 0.16(Re)^{-1/8}$ and TKE by the square of mean velocity. The third parameter, wall shear stress is normalized by its circumferentially averaged value.

4.1 Effects of Reynolds number

Effect of Reynolds number on turbulent intensity, TKE and wall shear stress for different cases are presented



(a) (b)

Fig. 2. Comparison between the present and published experimental (from Feng *et al.* 2014; Kim *et al.* 2014), numerical (from Kim *et al.* 2014; Tanaka *et al.* 2009) results (a) Relative total pressure value along the inner and outer core; (b) Normalized axial velocity distribution profile at bend outlet position.

in Fig. 3, Fig. 4, and Fig. 5 respectively. Although the present work was done for all three different curvature ratios, but here results are presented for the lower and higher curvature ratio cases only in order to observe the effect of Reynolds number. The horizontal central line at the bend outlet position has been considered as a common location to study turbulent intensity and turbulent kinetic energy. Turbulent intensity is defined as $I \equiv u'/U$, also often referred to as turbulence level. Where u´ and U are the root-mean-square of the turbulent velocity fluctuation and U is the mean velocity respectively.

From Fig. 3 it has been observed that the maximum value of turbulent intensity was found in the near-wall region (inner and outer) and comparatively minimum at the center region for simple pipe bend case. For lower curvature ratio (Rc/D = 1) and lower Reynolds number (1×10^5), the turbulent intensity at the inner curvature side shows two sharp peaks whereas for higher Reynolds number only one peak value is observed at the inner wall side. For higher curvature ratio (Rc/D = 2), there are also two peaks observed at the inner side but the maximum value of intensity is lesser compare to the lower curvature ratio (Rc/D = 1). Turbulent intensity near the inner and outer wall at bend outlet position show comparatively higher value because of the bend curvature. During the flow, as the curvature tries to change the flow direction, the fluid elements near the inner and outer wall become more unsteady and fluctuating in nature thus increases the value of turbulent velocity fluctuation.

Changes in turbulent intensity due to Reynolds number change is marginal at the outer wall side and the central region. There is a sudden increase in turbulent intensity observed at the location where the guide vane is placed. This is happening due to the presence of guide vane which causes a sudden change in the flow field and becomes a reason behind the high value of turbulent intensity. In general, Reynolds number is found as a weak function on turbulent intensity near the outer wall and central region while a little impact is found near the inner curvature wall. Changes in the central region were only observed in guide vane cases. After getting an overview of turbulent intensity, the effect of Reynolds number on turbulent kinetic energy has been studied.

The same horizontal central line position at the bend outlet has been taken as the common position for all the cases to represent the present analysis results on TKE. Turbulent kinetic energy denoted by the value of average kinetic energy per unit mass is about eddies in a turbulent flow system. The TKE is depending on the value of the estimated Root-Mean-Square (RMS) velocity fluctuation. Incorporating the Reynolds-averaged Navier-Stokes equations, the amount of turbulent kinetic energy is calculated with the help of the turbulence model. In general, the amount of turbulent kinetic energy can be estimated by determining the average of the turbulent normal stresses. Figure 4 show the impact of Reynolds number on TKE for all the cases and curvature ratio 1 and 2. Similar to the turbulent intensity, TKE is also found maximum towards the inner wall for a lower curvature ratio. Results for the other two Reynolds number show that the TKE distribution at the bend outlet position along the horizontal centerline is very much similar in nature. From the results, minimal changes have been observed due to the Reynolds number change.

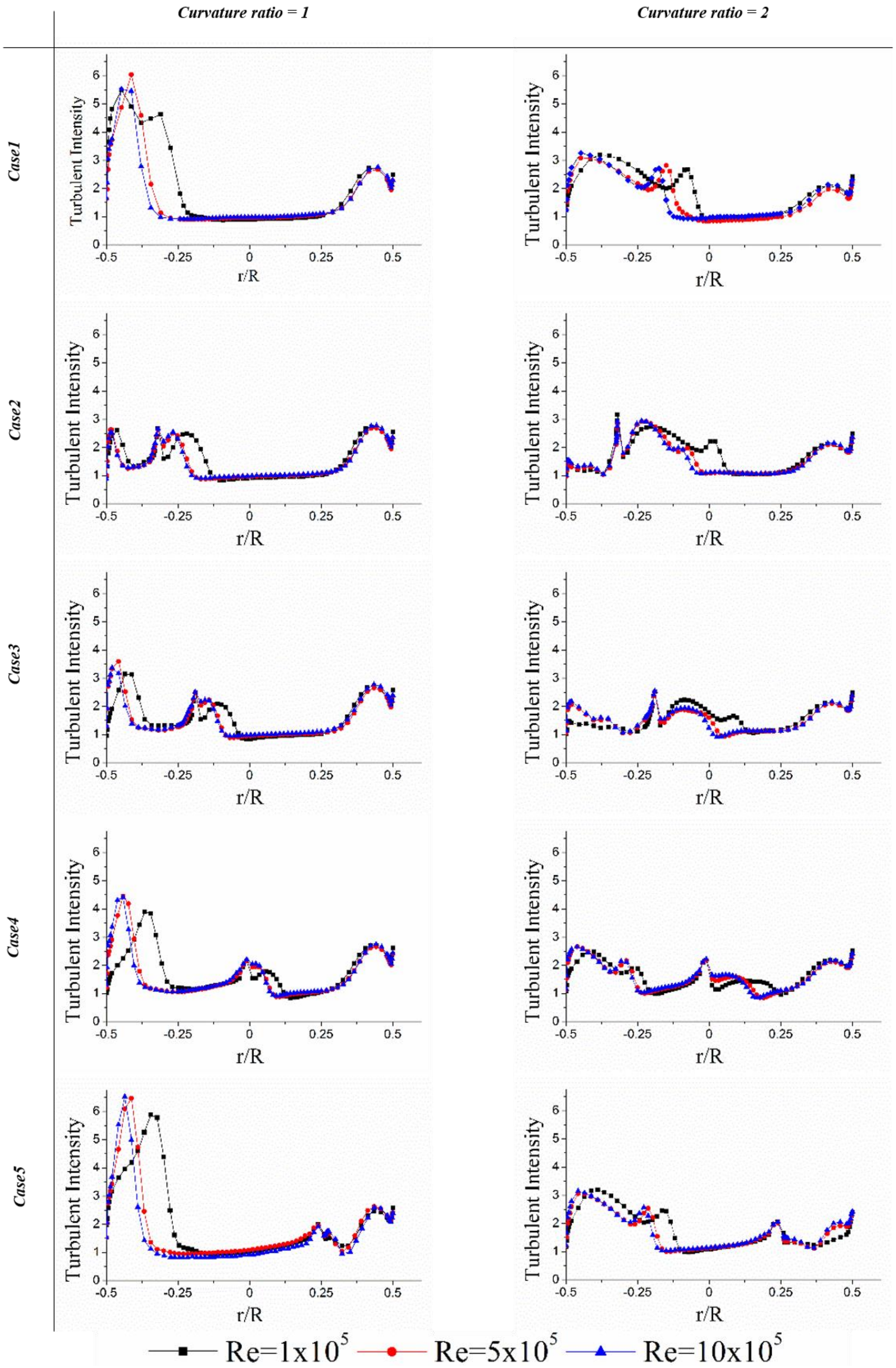


Fig. 3. Effect of Reynolds number on Turbulent Intensity at bend outlet position along the horizontal centerline from inner to outer curvature wall side (r/R value -0.5 represents the inner side and 0.5 as outer side).

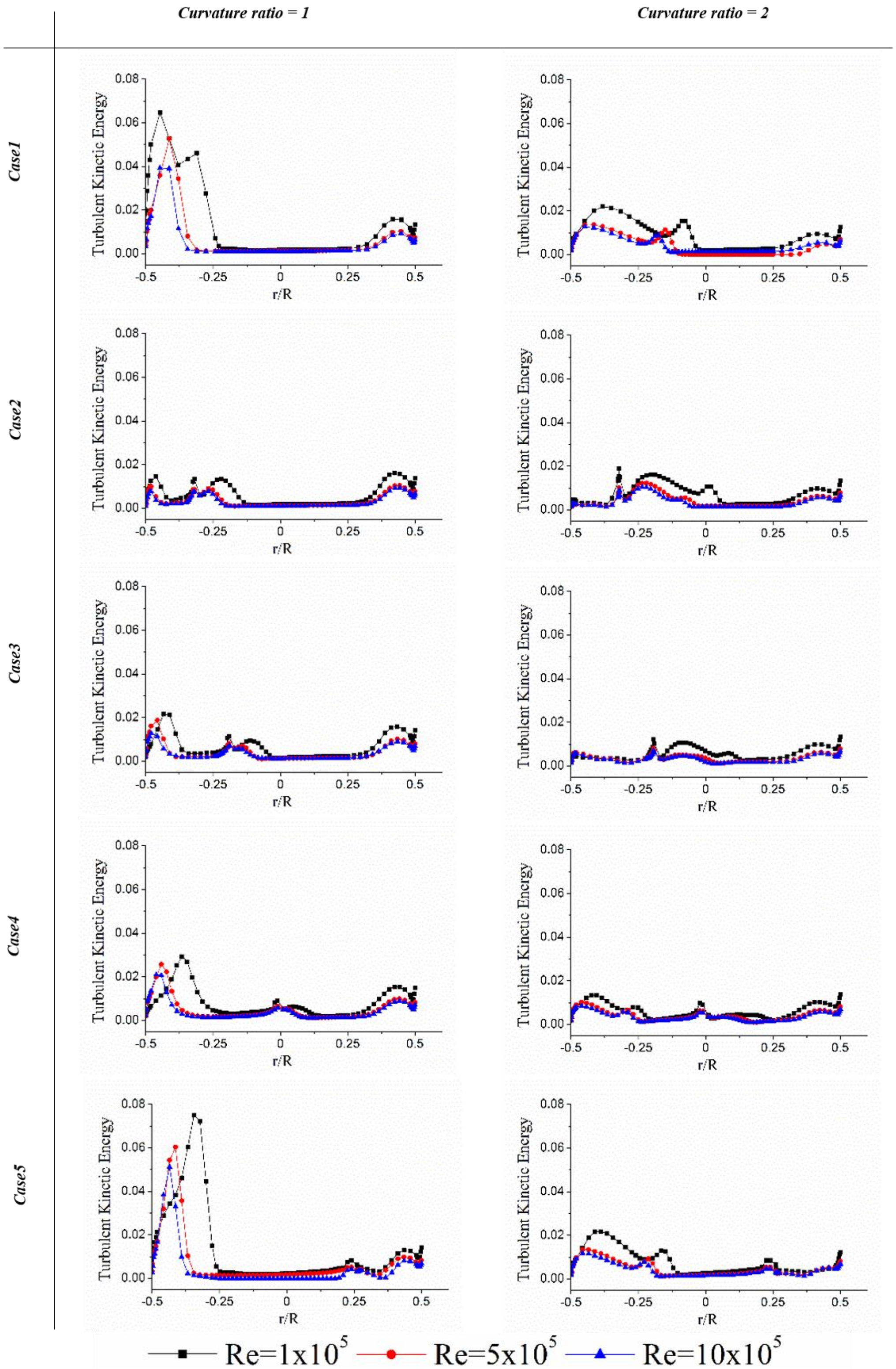


Fig. 4. Effect of Reynolds number on Turbulent Kinetic Energy at bend outlet position along the horizontal centerline from inner to outer curvature wall side (r/R value -0.5 represents the inner side and 0.5 as outer side).

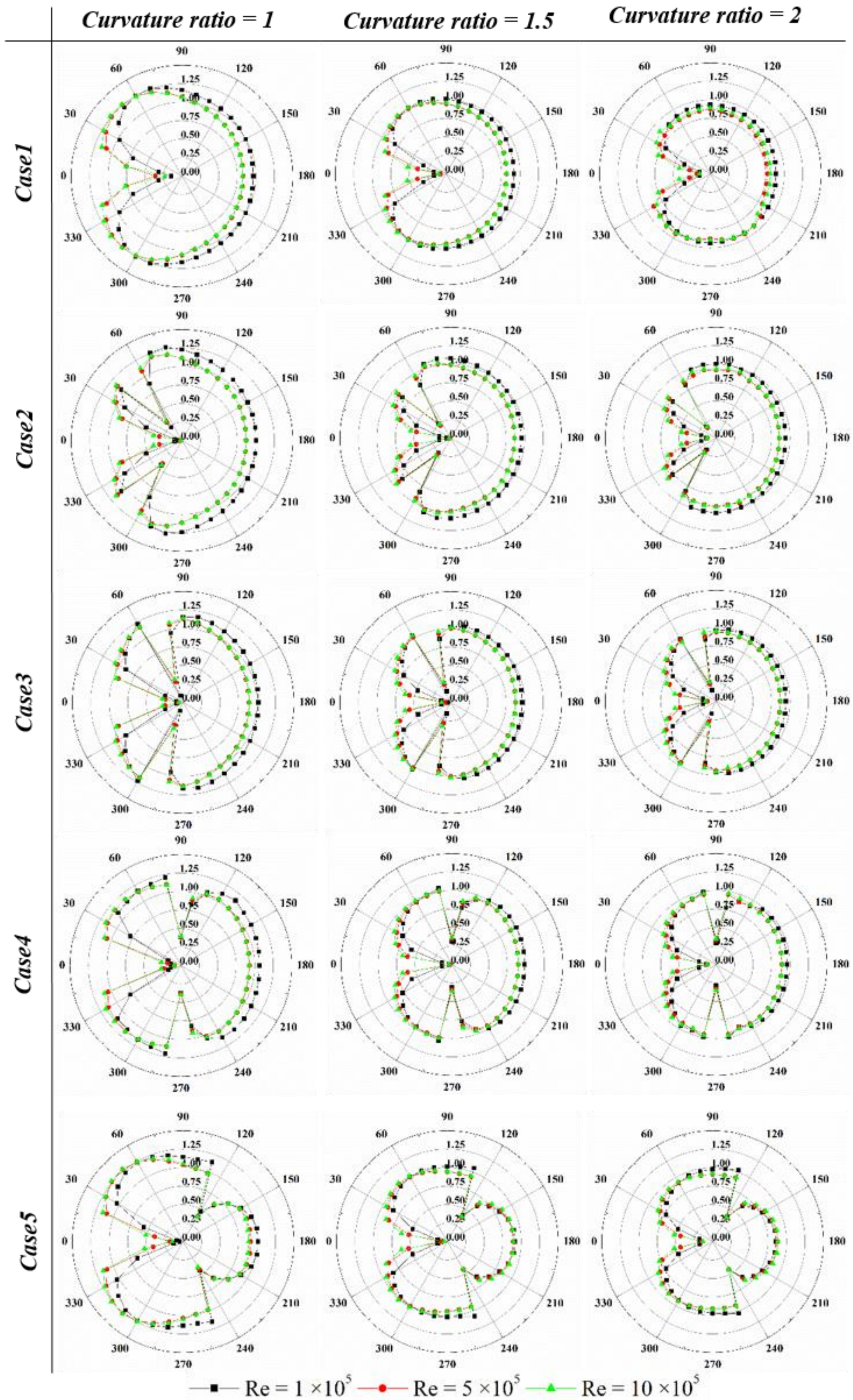


Fig. 5. Effect of Reynolds number on wall shear stress at bend outlet position.

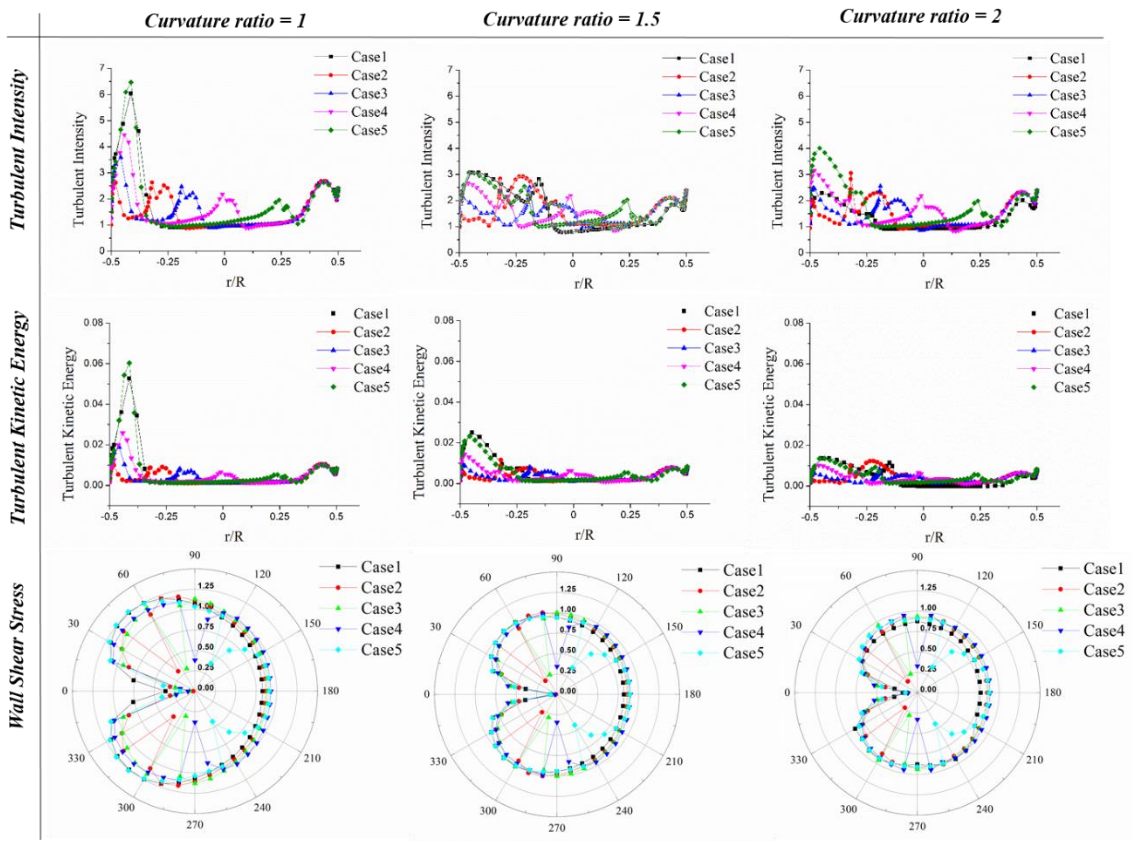


Fig. 6. Effect of Guide vane and its positions on (i) Turbulent Intensity. (ii) Turbulent Kinetic Energy, and (iii) Wall shear stress at bend outlet position.

It is well known that the flow accelerated corrosion frequently seemed near the high wall shear stress region of the flow field. Hence it is important to study the effect of different flow parameters (Reynolds number, curvature ratio, guide vane position) on wall shear stress distribution. Figure 5 shows the wall shear stress distribution at the bend outlet along the periphery for three different Reynolds number. In these polar diagrams, 0° indicates the inner wall side and 180° indicates the outer wall side. From Fig. 5, it has been noticed that the outer wall experiences comparatively high wall shear stress value than the inner wall region. As the Reynolds number increases, wall shear stress slightly increases near the inner curvature wall side (near $\theta = 30°$) and decreases at the outer wall region ($\theta = 180°$).

The study has been carried out with different Reynolds number corresponding to different positions of the guide vanes for different curvature ratios. It is observed that the Reynolds number is less influential on the three parameters (turbulent intensity, turbulent kinetic energy, and wall shear stress) in this present study. So, further results regarding the effect of Guide vane positions and Curvature ratios are presented here for a fixed Reynolds number (5×10^5) for the sake of brevity.

4.2 Effect of Guide vane position

To study the effect of guide vane and its position on turbulent quantities, a simple pipe bend (Case1, without guide vane) and four cases with guide vane (Case2 to Case5) at four different positions inside the pipe bend has been considered as mentioned in table 1. Figure 6 shows the effect of guide vane and its positions on turbulent intensity, turbulent kinetic energy, and wall shear stress for three different curvature ratios and one fixed Reynolds number 5×10^5. From the Fig. 6, it is clear that the normalized turbulence intensity and TKE have a major influence for guide vane and its position compare to normalized wall shear stress. Changes in the turbulent intensity and turbulent kinetic energy depend on turbulent velocity fluctuation where wall shear stress interpret the force on pipe wall per unit area. The guide vane inside the pipe bend mainly affects the flow field and changes the value of velocity fluctuations. So the guide vane and its positions are more influential for the changes in the normalized turbulent intensity and turbulent kinetic energy compare to normalized wall shear stress. The turbulent intensity and TKE are found maximum for Case1 (without guide vane) and Case5 (guide vane near to the outer curvature wall) at the inner wall for all three different curvature ratios.

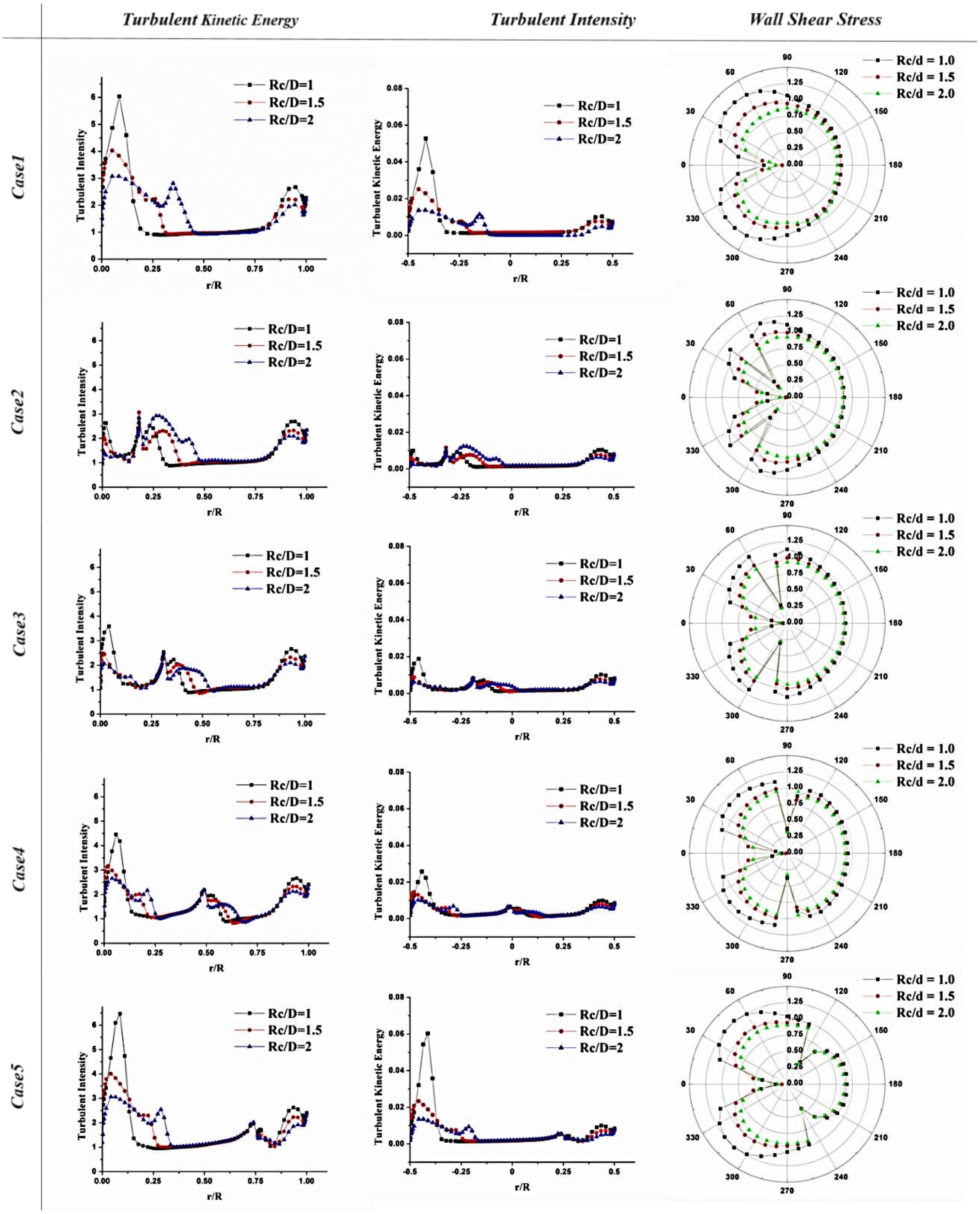


Fig. 7. Effect of Curvature ratio on (i) Turbulent Intensity. (ii) Turbulent Kinetic Energy, and (iii) Wall shear stress at bend outlet position

The minimum value of turbulent intensity has been found for Case2 (guide vane is placed near the inner curvature wall) near the inner wall. Similar type results were also observed for turbulent kinetic energy. It has been also observed that both turbulent intensity and TKE have a comparatively higher value at the inner core and outer core positions for Case1 and Case5. In between the inner and outer core, the value of turbulent intensity and TKE tries to gain minimum value but due to the guide vane effect, the value of turbulent intensity and TKE slightly increases at that particular position where the guide vane is placed inside the bend.

The turbulent intensity and TKE value near the inner wall side are minimum for Case2 (guide vane position near ICW side) and as the guide vane position shifts towards the OCW side, both the value of turbulence intensity and TKE increases gradually. Polar diagram plot for wall shear stress due to the effect of guide vane positions show that the overall distributions are almost the same along the periphery. There is a certain drop of wall shear stress value at the inner curvature side (θ = 0°) for both with and without guide vane cases. Wall shear stress value falls suddenly at different angular positions according to the location of guide vane.

4.3 Effect of curvature ratios

From Fig. 7, it is observed that as the curvature ratio increases, turbulent intensity and TKE decreases at both inner and outer wall side. This change is maximum at the inner wall side compared to the outer wall side. At the central region, there are no major changes noticed due to curvature ratio changes.

Wall shear stress is found very low at the inner side where generally flow separation occurs and the particular position where the guide vanes are positioned. Maximum shear stress is observed at the angular distance location between 30° to 60° and 300° to 330°. Wall shear stress is also decreased as the value of curvature ratio increases. The effect of curvature ratio on wall shear stress is comparatively high at the inner curvature wall region and low at the outer curvature wall region (θ = 90° to 270°).

5. CONCLUSIONS

The purpose of the study was to observe the change in turbulence characteristics due to different Reynolds number and geometrical modifications (*i.e.* curvature ratio & position of the guide vane) using the most reliable turbulence modelling i.e. SST k-ω turbulence. This physical situation has not been investigated earlier, neither by lab experiments nor by numerical simulation.In the present study, three different turbulence parameters have been taken into consideration, these are (i) Turbulent Intensity, (ii) Turbulent Kinetic Energy (TKE), and (iii) Wall shear stress. This study gives an overall idea about the presence and positional effect of guide vane on turbulence in the flow field which can be helpful for the engineers as well as researchers in the design of the piping network and mechanical devices to increase performance. The following conclusions can be made from the study.

1. The changes in turbulent intensity, turbulent kinetic energy measured along the horizontal centerline, and wall shear stress measured along the periphery of the circular cross-section due to change of Reynolds number have minimal effect near the inner wall region and negligible near the outer wall and central region at bend outlet.
2. The guide vane positions are more influential on turbulence parameters for lower curvature ratio (Rc/D = 1) cases than the medium (Rc/D = 1.5) and higher curvature ratio (Rc/D = 2) cases.

REFERENCES

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

ANSYS (2016), 'ANSYS Fluent - *CFD Software* | ANSYS'.

Anwer, M., R. M. C. So and Y. G. Lai (1989). Perturbation by and recovery from bend curvature of a fully developed turbulent pipe flow. *Physics of Fluids A: Fluid Dynamics* 1(8), 1387-1397.

Azzola, J., J. A. C. Humphrey, H. Iacovides and B. E. Launder (1986). Developing turbulent flow in a U-bend of circular cross-section: measurement and computation. *Journal of fluids engineering* 108(2), 214-221.

Di Liberto, M., I. Di Piazza and M. Ciofalo (2013). Turbulence structure and budgets in curved pipes. *Computers & Fluids* 88, 452-472.

Di Piazza, I. and M. Ciofalo (2011). Transition to turbulence in toroidal pipes. *Journal of Fluid Mechanics* 687, 72-117.

Dutta, P., S. K. Saha, N. Nandi and N. Pal (2016). Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling. *Engineering Science and Technology, an International Journal* 19(2), 904-910.

Enayet, M. M., M. M. Gibson, A. M. K. P. Taylor and M. Yianneskis (1982). Laser-Doppler measurements of laminar and turbulent flow in a pipe bend. *International Journal of Heat and Fluid Flow* 3(4), 213-219.

Feng, B., S. Wang, S. Li., X. Yang and S. Jiang (2014). Experimental and numerical study on pressure distribution of 90 elbow for flow measurement. *Science and Technology of Nuclear Installations*.

Gorman, J. M., E. M. Sparrow, C. J. Smith, A. Ghosh, J. P. Abraham, R. Daneshfaraz and Rezazadeh A. Joudi (2018). In-bend pressure drop and post-bend heat transfer for a bend with a partial blockage at its inlet. *Numerical Heat Transfer, Part A: Applications* 73(11),743-767.

Hellström, F. and L. Fuchs (2007). Numerical computations of steady and unsteady flow in bended pipes. *AIAA Fluid Dynamics Conference and Exhibit*, 25-28.

Hellstrom, L. H. O., M. B. Zlatinov, A. J. Smits and G. Cao (2011). Turbulent pipe flow through a 90° bend. *In Seventh International Symposium on Turbulence and Shear Flow Phenomena*. Begel House Inc.

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

Kim, J., M. Yadav and S. Kim (2014). Characteristics of secondary flow induced by 90-degree elbow in turbulent pipe flow. *Engineering Applications of Computational Fluid Mechanics* 8, 229–239.

Kotb, N. A. E., M. R. Mokhtarzadeh-Dehghan and A. J. Ward-Smith (1988). A numerical study of laminar and turbulent flows in a two-dimensional bend with or without a guide vane. *International Journal for Numerical Methods in Engineering* 26, 245–262.

Lee, G. H., Y. D. Choi and S. H. Han (2007). Measurement of developing turbulent flow in a U-bend of circular cross-section. *Journal of Mechanical Science and Technology* 21(2), 348-359.

Liou, T. M., H. L. Lee and C. C. Liao (2001). Effects of guide-vane number in a three-dimensional 60-deg curved side-dump combustor inlet. *J. Fluids Eng.* 123, 211–218.

Menter, F. (1993). Zonal two equation kw turbulence models for aerodynamic flows. In *23rd Fluid Dynamics, Plasmadynamics, and Lasers Conference* p. 2906.

Menter, F. R., (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal* 32, 1598-1605.

Moujaes, S. F. and S. Aekula (2009). CFD predictions and experimental comparisons of pressure drop effects of turning vanes in 90 duct elbows. *Journal of Energy Engineering* 135, 119–126.

Noorani, A., G. K. El Khoury and P. Schlatter (2013). Evolution of turbulence characteristics from straight to curved pipes. *International Journal of Heat and Fluid Flow* 41, 16-26.

Nugroho, S. and A. A. Hidayatulloh (2016). Performance Analysis of The Effect on Insertion Guide Vanes For Rectangular Elbow 900 Cross Section. EMITTER *International Journal of Engineering Technology* 4, 358–370.

Patankar, S. V., V. S. Pratap and D. B. Spalding (1975). Prediction of turbulent flow in curved pipes. *Journal of Fluid Mechanics* 67(3), 583-595.

Pellegrini, M., H. Endo and H. Ninokata (2011). Numerical investigation of bent pipe flows at transitional Reynolds number. *Progress in Nuclear Energy* 53(7), 916-920.

Pruvost, J., J. Legrand and P. Legentilhomme (2004). 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.

Reynolds, W. C., (1987). Fundamentals of turbulence for turbulence modeling and simulation, *Stanford Univ CA Dept of Mechanical Engineering*,

Roache, Patrick J. (1994). Perspective: a method for uniform reporting of grid refinement studies. *Journal of Fluids Engineering (American Society of Mechanical Engineers)* 116, 405-413.

Roache, Patrick J. (1998). Verification of codes and calculations. *AIAA Journal* 36, 696-702

Rowe, M., (1970). Measurements and computations of flow in pipe bends. *Journal of Fluid Mechanics*, 43(4), 771-783.

Röhrig, R., S. Jakirlić and C. Tropea (2015). Comparative computational study of turbulent flow in a 90-pipe elbow. *International Journal of Heat and Fluid Flow* 55, 120-131.

Rütten, F., M. Meinke and W. Schröder (2001). Large-eddy simulations of 90-pipe bend flows. *Journal of Turbulence* 2, p. N3.

Spalart, P. R. (2000). Strategies for turbulence modelling and simulations. *International Journal of Heat and Fluid Flow* 21, 252-263.

Sparrow, E. M., J. C. Tong and J. P. Abraham (2008). Fluid flow in a system with separate laminar and turbulent zones. *Numerical Heat Transfer, Part A: Applications* 53(4), 341-353.

Speziale, C. (1998). Turbulence modeling for time-dependent RANS and VLES: a review. *AIAA Journal* 36, 173-184.

Sudo, K., M. Sumida and H. Hibara (1998). Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend. *Experiments in Fluids* 25(1), 42-49.

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

Sugiyama, H. and D. Hitomi (2005). Numerical analysis of developing turbulent flow in a 180 bend tube by an algebraic Reynolds stress model. *International Journal for Numerical Methods in Fluids* 47(12), 1431-1449.

Swienty, A., A. C. K. L. Raja Abou and P. U. Thamsen (2015). Assessing the accuracy of turbulence models for resolving shear and swirl flows in pipes. *The 16th International Conference on*

Fluid Flow Technologies Budapest, Hungary, September 1-4

Tanaka, M. A., H. Ohshima and H. Monji (2009). Numerical investigation of flow structure in pipe elbow with large eddy simulation approach. In *ASME Pressure Vessels and Piping Conference* 43666., 449–458.

Valsala, R. R., S. W. Son, A. Suryan and H. D. Kim (2019). Study on reduction in pressure losses in pipe bends using guide vanes. *Journal of Visualization* 22, 795–807.

Wilcox, D. C. a. o., (1998). Turbulence modeling for CFD. *DCW industries La Canada, CA*. Vol 2, 103-217

Zhang, T., Y. O. Zhang and H. Ouyang (2015). 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.

Archive of SID.ir