Research Article

Numerical Investigation on Fluid Flow in a 90-Degree Curved Pipe with Large Curvature Ratio

Yan Wang, Quanlin Dong, and Pengfei Wang

School of Instrumentation Science and Opto-Electronics Engineering, Beijing University of Aeronautics and Astronautics, Beijing 100191, China

Correspondence should be addressed to Yan Wang; wbybuaa@163.com

Received 10 May 2015; Revised 14 July 2015; Accepted 27 July 2015

Academic Editor: Francesco Pesavento

Copyright © 2015 Yan Wang et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

In order to understand the mechanism of fluid flows in curved pipes, a large number of theoretical and experimental researches have been performed. As a critical parameter of curved pipe, the curvature ratio δ has received much attention, but most of the values of δ are very small ($\delta < 0.1$) or relatively small ($\delta \leq 0.5$). As a preliminary study and simulation this research studied the fluid flow in a 90-degree curved pipe of large curvature ratio. The Detached Eddy Simulation (DES) turbulence model was employed to investigate the fluid flows at the Reynolds number range from 5000 to 20000. After validation of the numerical strategy, the pressure and velocity distribution, pressure drop, fluid flow, and secondary flow along the curved pipe were illustrated. The results show that the fluid flow in a curved pipe with large curvature ratio seems to be unlike that in a curved pipe with small curvature ratio. Large curvature ratio makes the internal flow more complicated; thus, the flow patterns, the separation region, and the oscillatory flow are different.

1. Introduction

Curved pipes have a very wide range of applications in industry, such as ventilation pipes, heat exchangers, and turbine machineries. In addition, in physiology, the physical models of curved pipes are very similar to those of blood vessels; many physiologists explain the flow pattern in those vessels by studying flow characteristics in curved pipes.

As a pioneer in research of fluid motion through curved pipes, based on the experiments of Eustice [1], Dean [2, 3] researched the incompressible fluid motion through a curved pipe with very small curvature ratio in a laminar flow environment, providing a theoretical solution of the flow streamline in Eustice's experiments and finding a secondary flow on the cross section of curved pipe. He defined a dimensionless number $K = 2\text{Re}^2 a/R$, which represents the impact of the characteristics of the fluid and the geometry of the curved pipes on the flow field, where a is the pipe cross section radius, R is the curvature radius, and Re is the Reynolds number. He represented that his analysis was only valid for small curvature ratios and $K \leq 576$. After this research, White [4] and Taylor [5], respectively, proved Dean's theory in their

experiments. Taylor also confirmed that the fluid in a curved pipe is more stable than that in a straight pipe. This result means that the critical Reynolds number of the former is greater in the same conditions. Subsequently McConaughy and Srivastava [6] supplemented and expanded Dean's research achievements by using the Fourier series expansion and defined a new dimensionless number $D = 4K^{1/2} = 4\text{Re}(2a/R)^{1/2}$. In their study, the Fourier series expansion was used successfully for $D < 106.07$. Greenspan [7] used finite difference method, expanding the range of D to the entire laminar flow based on previous research with small curvature ratios. A factored ADI finite-difference scheme had been used for numerical calculation on a curved pipe of arbitrary curvature ratio by Soh and Berger [8]. Authors had calculated three values of δ : 0.01, 0.1, and 0.2 in the range of $5 \leq D \leq 30000$; the results showed that both the fluid flow and the friction are greatly influenced by the value of δ .

Considering the fact that the detail and accuracy of the measurements were not enough, Taylor et al. [9] measured the flow velocity field in a square cross section 90-degree curved pipe with small curvature ratio ($\delta = 0.217$) by using Laser Doppler velocimetry under laminar and turbulent

environments. Later on, Sudo et al. [10, 11] provided detailed information on turbulent flow through a circular-sectioned and a square-sectioned 90-degree curved pipe with small curvature ratio ($\delta = 0.25$) by using the technique of rotating a probe with an inclined hot wire. The experiment involved curved and straight upstream and downstream pipes. In addition, they also studied the deviation of primary flow and intensity of secondary flow. The conclusions showed that there is no significant boundary layer separation in the bend. Although the effect of the curvature ratio was explained, the value of δ was restricted to small scales. The water experiments in two elbows with different curvature ratio ($\delta = 0.33$ and 0.5) were studied using a high-speed PIV by Ono et al. [12]. They found that the curvature ratio affects the continuity of separation region generation and the secondary flow affects the flow in the separation region. Tan et al. [13] evaluated the fluid flow in pipes with two different curvature ratios $\delta = 0.25$ and $\delta = 0.5$. The former had been experimentally investigated by Sudo et al. [10]. They found that the curvature has a considerable impact on the pressure and velocity distributions. A stronger flow separation would happen at the inner side of pipe with larger curvature ratio.

In the process of experiment, Tunstall and Harvey [14] noted that there is a unique secondary flow pattern in a sharp curved pipe with $Re = 4 \times 10^4$, which is different from the well-known secondary flow. This single swirl flow dominated the flow downstream of the bend in a clockwise or an anticlockwise direction and switched its direction abruptly at a long, random timescale. Subsequently, the different turbulence models and wall equations were used to investigate the fluid flow in the 90-degree and 180-degree bend with small curvature ratio ($\delta = 0.25$) by Pruvost et al. [15]. The results showed that the relation between swirl motion and Dean motion is complicated and swirl motion has an inhibitory effect against Dean motion. Rütten et al. [16] investigated turbulent flows through a 90° elbow by using LES, where the curvature ratios were 0.167 and 0.5, respectively. The authors focused on internal unsteady flow separation, unstable shear layers, and oscillation of Dean vortices. They confirmed swirl switching and boundary layer separation. Furthermore, they found that low-frequency oscillation variation is a smooth process rather than switching abruptly, and the low-frequency oscillation does not depend on the presence of flow separation. Hellström et al. [17] studied the curvature effect for the flow field in the downstream of the curved pipe with $\delta = 0.5$ by using PIV with Reynolds numbers between 2×10^4 and 1.15×10^5 . Combined with snapshot proper orthogonal decomposition (POD), they found that swirl switching has a more energetic structure than Dean motion. They further proposed that the fluid flow at the inner corner of the bend is greatly influenced by upstream.

In general, the investigations on fluid flows in 90-degree curved pipes are roughly divided into three types: theoretical analysis, numerical simulation, and experimental investigation. Although researchers have made great progress in the study of characteristics of fluid motions in curved pipes, because of the complexity and diversity of the flow field, there are still many problems to be further studied. In addition, researchers have mainly focused on curved pipes of small

FIGURE 1: Schematic representation of a 90-degree curved pipe with large curvature ratio.

curvature ratio or small Dean number which are important in biological applications and some industrial applications. Numerical and experimental studies for curved pipes with large curvature ratio ($\delta > 0.5$) are very few in the literature. Berger et al. [18] believed that a curved pipe of large curvature ratio might be different from that of small curvature ratio. In this paper, based on the previous literature, the fluid flows through a curved pipe for $\delta > 0.5$ are predicted by numerical simulation. Flow behaviors, such as secondary flow, boundary layer separation, and the oscillatory flow, are illustrated and studied. Furthermore, the variation of flow characteristics, such as the turbulence intensity and the secondary flow intensity, is estimated for a given flow condition.

2. Model and Numerical Method

The geometry size of model used in this work is shown in Figure 1. This paper assumes that the fluid is an incompressible air. The curved pipe inner diameter $d = 500$ mm, and a curvature radius $R = 410$ mm; therefore the curvature ratio $\delta = d/2R = 0.610$. Upstream and downstream tangents are 3000 mm ($z/d = -6$) and 1000 mm ($z'/d = 2$), respectively. θ is angle of cross section of curved pipe around O point, such as $\theta = 0^\circ$ ($z/d = 0$) which is at the entrance of curved pipe and $\theta = 90^\circ$ ($z'/d = 0$) which is at the exit of curved pipe; r and α are radial and circumferential coordinates of cross section.

Nobari and Rajaei [19] employed direct numerical simulation (DNS) for developing flow in a curved square annulus since DNS is the most appropriate method for turbulence research. However, in most cases, because of excessive consumption DNS is not often used in practical problems. Kuan and Schwarz [20] used the standard $k-\epsilon$ model and differential Reynolds stress model (DRSM) to study turbulent flows in bends. Compared with experimental data, the numerical results showed that it has a satisfactory performance for time average velocity before the $\theta = 45^\circ$ position. From $\theta = 45^\circ$ to $3d$ downstream of bend, there is considerable difference between experimental measurement and numerical calculation. Raisee et al. [21] used two different low Reynolds eddy viscosity models, a linear $k-\epsilon$ model and a nonlinear $k-\epsilon$ model, for the numerical prediction of the velocity and pressure fields in three-dimensional turbulent flow field through curved pipes. According to the conclusions, both models could show satisfactory prediction of the mean flow field. The nonlinear $k-\epsilon$ model has better performance for turbulence

field and pressure and friction coefficients, but it is not accurate for the prediction of flow recovery after the bend exit. Compared with DNS results and experimental data, Di Piazza and Ciofalo [22] attempted to evaluate the predictive ability of turbulence models (k - ε , SST k - ω , and RSM- ω). They found that SST k - ω and RSM- ω models agree very well with experimental data and the latter is slightly better in predicting the details of velocity and temperature profiles. Berrouk and Laurence [23] suggested that based on experimental data LES is more effective in predicting fluid flow than Reynolds Averaged Navier-Stokes (RANS) models. The same conclusion was given by Zhang et al. [24] and Tan et al. [13]. Previous studies have indicated that LES is one of the most appropriate turbulence models to predict flows in curved pipes.

Compared with DNS and RANS, LES is a compromise approach. It can obtain details on the structure of transient flow. This method is to separate large-scale and small-scale transient fluctuation motions. Large-scale transient fluctuation motions are solved directly by the Navier-Stokes equations, while small-scale motions are calculated implicitly by subgrid scale model (SGS model).

For incompressible flow, the equations $u_i = \bar{u}_i + u'_i$ and $p_i = \bar{p}_i + p'_i$ are substituted into the continuity equation and the Navier-Stokes equations, yielding the filtered incompressible continuity equation and Navier-Stokes equations:

$$\begin{aligned} \frac{\partial \bar{u}_i}{\partial x_i} &= 0, \\ \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) &= -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right], \end{aligned} \quad (1)$$

where u_i and p_i are the fluid transient variables, $\bar{u}_i$ and $\bar{p}_i$ are the unresolved variables, and u'_i and p'_i are the subgrid-scale variables. The term $\bar{u}_i \bar{u}_j$ can be written as

$$\bar{u}_i \bar{u}_j = \bar{u}_i \bar{u}_j + (\bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j) = \bar{u}_i \bar{u}_j + \tau_{ij}. \quad (2)$$

Therefore the following governing equations can be obtained:

$$\begin{aligned} \frac{\partial \bar{u}_i}{\partial x_i} &= 0, \\ \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) &= -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] \\ &\quad - \frac{\partial \tau_{ij}}{\partial x_j}. \end{aligned} \quad (3)$$

In (4), it seems that the expression is very similar to RANS, where τ_{ij} represents the subgrid-scale stress; it is generated from nonlinear convective terms during filtering, expressed as the small-scales impact on large-scales. The relationship between subgrid-scale stress τ_{ij} and the large-scale strain rate tensor $\bar{S}_{ij}$ is defined as

$$\tau_{ij} = \frac{\delta_{ij}}{3} \tau_{kk} - 2\nu_{\text{sgs}} \bar{S}_{ij}, \quad (5)$$

where $\bar{S}_{ij}$ is the strain rate tensor for the resolved scale and the subgrid-scale viscosity ν_{sgs} represents the small scales that are defined as

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right). \quad (6)$$

Compared to the Smagorinsky model, the wall-adapting local eddy-viscosity (WALE) model by Nicoud and Ducros [25] is based on the square of the velocity gradient tensor; therefore the effects of both the strain and the rotation rate of the smallest resolved turbulent fluctuations are considered. This model is also capable of dealing with the laminar to turbulent transition. Moreover it needs no dynamic procedure to recover the correct wall-asymptotic y^3 -variation of the SGS viscosity. In the LES WALE model the eddy viscosity ν_{sgs} is defined as

$$\nu_{\text{sgs}} = (C_w \Delta)^2 \frac{(S_{ij}^d)^{3/2}}{(\bar{S}_{ij} \bar{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}}, \quad (7)$$

$$S_{ij}^d = \frac{1}{2} (\bar{g}_{ij}^2 + \bar{g}_{ji}^2) - \frac{1}{3} \delta_{ij} \bar{g}_{kk}^2,$$

where the constant C_w is set to 0.5 [25], the velocity gradient tensor $\bar{g}_{ij}$ denotes $\partial \bar{u}_i / \partial x_j$, $\bar{g}_{ij}^2$ denotes $\bar{g}_{ik} \bar{g}_{kj}$, and δ_{ij} is the Kronecker symbol.

For LES turbulence model, if the whole flow field is solved by this model, the cost of computation is too large. This work adopted Detached Eddy Simulation (DES) method; therefore the whole flow field was divided into the near-wall flow region and the core flow region. LES was applied in the core flow region and the Omega-based model was employed in the near-wall flow region.

The ω -equation is in the sublayer and logarithmic region as follows:

$$\begin{aligned} \omega_{\text{sub}} &= \frac{6\nu}{\beta (\Delta y)^2}, \\ \omega_{\text{log}} &= \frac{u^{*2}}{c_1 \kappa \nu y^+}, \end{aligned} \quad (8)$$

where β and c_1 are constant, Δy is the distance between the first and the second mesh point, ν is kinematic viscosity, κ is the von Karman constant, y^+ is the dimensionless distance from the wall, and u^* is the velocity scale in the logarithmic region. And then friction velocity for the sublayer and logarithmic region is given as follows:

$$u_\tau = \sqrt[4]{(u_\tau^{\text{vis}})^4 + (u_\tau^{\text{log}})^4}, \quad (9)$$

where $u_\tau^{\text{vis}} = u^+ / y^+$, $u_\tau^{\text{log}} = u^+ / (1/\kappa \log(y^+) + C)$, C are constant, and u^+ is the near-wall velocity.

3. Mesh and Boundary Conditions

In order to exclude the impact of grid numbers, the grid numbers ranging from 0.46×10^5 to 15.53×10^5 were generated

FIGURE 2: Pressure coefficient changes with the grid at different Dean numbers.

by ICEM and tested by CFX. According to the literature [13, 16], a reasonable magnitude of the grid number can be got. Based on their work, grid independence checks were carried out. Figure 2 shows that the pressure coefficient C_p changes with meshes at different Dean numbers ($De = K^{1/2}$). The pressure coefficient is defined as $C_p = 2(p - p_{ref})/\rho v_m^2$, where p_{ref} is the pressure at $z'/d = -4$, ρ is density of air, and v_m is mean velocity at the pipe inlet.

The boundary conditions are given as follows:

- (1) Inlet conditions: the inlet flow rate v_{in} is decided by Reynolds number ($5 \times 10^3 \leq Re \leq 2 \times 10^4$), and the direction is normal to the inlet. The velocities u and v both are zero on the remaining two directions. The ambient temperature is 25°C .
- (2) Outlet conditions: the opening boundary condition is chosen which means that the fluid is allowed to cross the boundary surface in either direction.
- (3) Wall conditions: no-slip wall assumes that relative velocity is zero between the surface and the gas, which means that $u = v = w = 0$ at the surface. The wall roughness is smooth, and the temperature of boundary condition is T_w , which means that the wall temperature is denoted by T_w and the temperature of the fluid layer in contact with the wall is also denoted by T_w .

The discretization algorithm for the transient term adopted the Second Order Backward Euler scheme. The time step $\Delta t = 0.05$ s and total time $T = 60$ s.

Figure 2 shows that the gap between the various pressure coefficients is not very obvious when the number of grid is greater than 1.69×10^5 . In this paper, the grid of 12.38×10^5 is a sample for analysis. Meshes topology is shown in Figure 3.

4. Simulations and Result Analysis

In this section, according to the previous numerical method and boundary conditions, fluid flows through the curved pipe with large curvature ratio were simulated and analyzed for $Re = 5000$ to 20000 by ANSYS CFX. This section has two components. Firstly, pressure field study is performed for the impact of large curvature ratio at different Reynolds numbers. On the other hand, the fluid flow and flow structure along the curved pipe are studied.

4.1. Pressure Distribution and Pressure Drop along the Curved Pipe. As shown in Figure 4, the pressure coefficient C_p is plotted at various positions of the pipe, where p is the pressure at wall. The geometry structure is symmetrical from upper to lower, so the calculation results in the lower half of the cross section of pipe are shown in the picture. The tendency of the pressure coefficient for the large curvature ratio is similar to that for the small one by Sudo et al. [10], Pruvost et al. [15], and Tan et al. [13]. But the difference is that the pressure coefficient does not show obvious peak at the outer wall, and with different Reynolds numbers the change of the outer wall pressure coefficient is minimized at the three positions. Compared with the experimental data [10], the calculation results show that the curvature ratio has a greater impact on pressure at the pipe inner side than at the pipe outer side.

Figure 5 gives the pressure coefficient gradient distribution along the pipe. The pressure coefficient gradient can be expressed as $\partial C_p / \partial r$. With the same Dean number, the calculation results agree well with the data [13]. In order to estimate the effects of centrifugal force, the forces on a gas element are shown in detail in Figure 6. Here the gas element is moving through the curved pipe. There are two accelerations: one is dw/dt along the tangential direction of the streamline and another is the centripetal acceleration w^2/R along the radius of curvature. These inertia forces and pressure on the element are in equilibrium, and the following equations are obtained:

$$-\frac{\partial p}{\partial n} + \rho \frac{w^2}{R} = 0, \quad (10)$$

$$-\frac{\partial p}{\partial s} - \rho \frac{dw}{dt} = 0. \quad (11)$$

According to (10), we find that the pressure gradient along the $-n$ direction is caused by centrifugal force due to the fluid flow in the curved pipe. On the other hand, it is shown that the flow in the upstream and downstream pipes is also influenced by centrifugal force in Figure 5. The pressure gradient decreases initially and increases afterwards before the entrance of curved pipe. The maximum pressure gradient appears at $\theta = 45^\circ$ with $\delta = 0.25$ [10]. According to the calculation results, the peak pressure coefficient gradient appears at $\theta = 30^\circ$ in the curved pipe with larger δ , which appears earlier than in the curved pipe with smaller δ . Thereafter the pressure gradient is decreasing because the centrifugal influence begins to weaken. Between $z'/d = 1$ and 2 , the pressure gradient decreases initially and increases afterwards again due to the influence of inertia force.

FIGURE 3: Computational grid of curved pipe with large curvature ratio.

FIGURE 4: Distributions of pressure coefficient along the pipe.

Figure 7 shows the distribution of the pressure loss coefficient h on each cross section, where l is the distance of the pipe centerline from the entrance to the current cross section, the zero position is the entrance of the curved pipe ($z/d = 0$), and the curved section is located between 0 and 1.29. The pressure loss in the straight part of pipe is only caused by friction. In the curved part, fluid flow is disturbed by secondary flow and the flow direction is changed; therefore the pressure loss is greater than the straight part. Similar decreasing trend is obtained in the research of Ito [26] and Spedding et al. [27]. According to (11), we find that the fluid has slightly decelerated initially and accelerated afterwards, and the acceleration process continues until $z'/d = 2$. The maximum value of the pressure loss coefficient of the bending

FIGURE 5: Pressure coefficient gradient along the pipe.

FIGURE 6: Forces on a gas element in the curved pipe.

is at $\theta = 30^\circ$. As the Reynolds number increases, the peak becomes more pronounced.

In order to observe the phenomenon of oscillatory flow at the curved pipe wall, the pressure coefficient C_p changes with time at different observation points are performed as in

FIGURE 7: Distribution of pressure loss coefficients along the pipe centerline.

FIGURE 8: Variation of the pressure coefficient with time at $Re = 10000$, monitoring point 1 at inner wall at $z/d = -1$, monitoring point 2 at outer wall at $z/d = -1$, monitoring point 3 at inner wall at $z'/d = 1$, and monitoring point 4 at outer wall at $z'/d = 1$.

Figure 8. At $z/d = -1$, there are the same pressure fluctuation curves at the inner and outer walls of the upstream curved pipe, the pressure coefficient at point 2 is greater than that at point 1, and the difference is maintained at around 0.045 with time from $T = 17$ s to 60 s. At $z'/d = 1$, the result shows that, after oscillatory flow impacts on the wall pressure, the pressure fluctuation curves are no longer synchronized, the phase difference is about 0.25 s, and the pressure coefficient at the outer wall is less than that at the inner wall. Due to the boundary layer separation of the downstream curved pipe, the difference of the pressure coefficient between point 3 and point 4 is not a fixed value. Moreover, as time increases, the amplitude of the pressure fluctuation will gradually weaken.

4.2. Fluid Motion and Flow Structure in the Curved Pipe. Distributions of axial velocity on each section with $Re = 5000, 10000$, and 20000 are plotted as shown in Figure 9, W is the axial velocity at the current position, and W_0 is the average axial velocity at the inlet. At $z/d = 1$, the fluid is not affected by the curvature of the curved pipe, and the inner and outer

axial velocity maintain a symmetrical structure which is the same as Poiseuille flow. Thereafter the axial velocity profile on the cross section appears to have a nonsymmetrical structure. From $\theta = 0^\circ$ to 60° , the axial velocities near the inner wall are faster than the axial velocities near the outer wall. In addition, before $\theta = 30^\circ$, the fluid is accelerated by an adverse pressure gradient in the axial direction near the inner wall, and the fluid is decelerated by a positive pressure gradient in the axial direction near the outer wall.

Due to the effect of centrifugal force, the boundary layer at the inner wall is getting thicker and the boundary layer at the outer wall is getting thinner. When the centrifugal force continues to increase, the boundary layer will separate from the inner wall. What is more, from $\theta = 60^\circ$ to $z'/d = 1$, this region generates reflux. These factors significantly decrease the axial velocity, and the axial velocity reaches the minimum near the inner wall region at $\theta = 90^\circ$. When the Reynolds number increases, it can be seen that the impact region of the reflux is gradually reduced. Compared with the literature [10, 13], although the Dean numbers are much smaller in

FIGURE 9: Distribution of axial velocity in each cross section for (a) $Re = 5000$, (b) $Re = 10000$, and (c) $Re = 20000$.

this paper, the internal flow field is much more complicated. At downstream $z'/d = 2$, the distribution of axial velocity has not yet recovered from centrifugal influence. It needs more distance to return to the distribution of axial velocity at upstream $z'/d = 1$. If the downstream length is not long enough, it will form a jet-wake structure at outlet of the pipe.

As shown in Figure 10, at $\theta = 0^\circ$, a secondary flow moves to the inner side due to the influence of curvature. From $\theta = 30^\circ$ to 60° , due to the centrifugal force and viscous force, the secondary flow gradually develops and it forms two opposite vortices in the cross section, having a similar manner to curved pipes with small curvature ratio [28]. The vortices circulate outwards near the center of the pipe and inwards near the upper and lower walls. Therefore the faster fluid near the inner side in the cross section is accompanied by this secondary flow gradually moving to the outer wall. At $\theta = 45^\circ$, the centers of two vortices both are skewed towards the inner

wall. At $\theta = 60^\circ$, the boundary layer begins to separate from the inner wall; therefore a gap occurs between the boundary layer and the inner wall, and the mainstream begins to reflux. This fluid motion is more obvious in the curved pipe as shown in Figure 11. At $\theta = 90^\circ$, there are a pair of deputy vortices to be generated near the inner wall region as shown in Figures 10(a) and 10(b). It also can be seen that the impact region of the reflux is gradually reduced with Reynolds number increase. What is more, the energy transfer and consumption would accelerate in the pipe since the mainstream and secondary flows have combined to produce complex spiral flow.

Velocity contours in center section with different Reynolds numbers are presented in Figure 11. At $\theta = 60^\circ$, due to the impact of the boundary layer separation and centrifugal force, the mainstream deflection is towards the outer side and thus the fluid near the outer wall is squeezed near the entrance of the curved pipe, which causes a vortex

FIGURE 10: Continued.

FIGURE 10: Secondary flow in each cross section for (a) $Re = 5000$, (b) $Re = 10000$, and (c) $Re = 20000$.

to appear near the outer side of the center section near the entrance of the curved pipe. As the Reynolds number increases, it can be seen that the boundary layer separation point gradually moves backward.

Figure 12 shows the turbulence intensity I_t and the secondary flow intensity I_s at different Reynolds numbers in curved pipe with large curvature ratio, where I_t and I_s are written as

$$I_t = \frac{8}{\pi d^2 v_m} \int_{-\pi/2}^{\pi/2} \int_0^{d/2} \sqrt{\frac{1}{3} (u'^2 + v'^2 + w'^2)} r dr d\alpha, \quad (12)$$

$$I_s = \frac{8}{\pi d^2 v_m^2} \int_{-\pi/2}^{\pi/2} \int_0^{d/2} (\bar{U}^2 + \bar{V}^2) r dr d\alpha,$$

where $\bar{U}$, $\bar{V}$, $\bar{W}$ are time mean velocities and u' , v' , w' are root mean square velocity (RMS) fluctuations in r , α , θ directions, respectively.

The turbulence flow and secondary flow intensities both have relatively strong fluctuations at the bend inlet and outlet. At $\theta = 0^\circ$, the differences of I_s in value for the pipe at different Reynolds numbers are very small. In contrast, the differences

are apparent at $\theta = 90^\circ$. Normally, for fully developed pipe flow I_t can be estimated as

$$I_t = 0.16 Re^{-1/8}. \quad (13)$$

It is shown that I_t is normally in inverse ratio to Re . As shown in Figure 12(a), this conclusion is correct, but due to the influence of curvature I_t is not a stable value. As θ increases, the turbulence flow I_t increases, and the maximum of I_t would be about 4 times the calculated value by (13). The intensities of turbulence flow and secondary flow become the strongest at $\theta = 90^\circ$, respectively.

5. Conclusions

The turbulence flows in a 90-degree curved pipe with large curvature ratio are performed at the Reynolds number range from 5000 to 20000 using the commercial code ANSYS CFX. The pressure and velocity distribution, pressure drop, oscillatory flow, and secondary flow along the curved pipe are studied in this work in order to analyze the flow characteristics. The conclusions obtained in this paper are summarized as follows:

FIGURE 11: Velocity contours in center section of pipe for (a) $Re = 5000$, (b) $Re = 10000$, and (c) $Re = 20000$.

- (1) Compared with a curved pipe of small curvature ratio, at $\theta = 30^\circ$ to 60° , we observed that the pressure change is relatively stable at the bend outer side with large curvature ratio. The curvature ratio has a great impact on pressure distribution, especially the pressure at the pipe inner side. As the curvature increases, the peak pressure coefficient gradient appears at $\theta = 30^\circ$.
- (2) Centrifugal force not only affects the pressure distribution in the curved section of pipe, but also has an impact on the pressure distribution in the upstream and downstream pipes. Therefore the pressure gradient has small fluctuations near the bend inlet and outlet.
- (3) The impacts of the oscillatory flow on the inner wall and outer wall are different, and the pressure coefficient at the outer wall is less than that at the inner wall in the downstream curved pipe. Because of the boundary layer separation, the difference of the pressure coefficient between point 3 and point 4 is not a fixed value, which is different from the upstream curved pipe. As time increases, the amplitude of the pressure fluctuation will gradually weaken.
- (4) As curvature ratio increases, the boundary layer separation becomes more obvious after $\theta = 60^\circ$. It generates a lot of reflux in the separation region. Meanwhile, due to the impact of the boundary layer separation and centrifugal force, a vortex appears near

FIGURE 12: Turbulence intensity and secondary flow intensity along the curved pipe.

the outer side of the center section near the entrance of the curved pipe.

- (5) For the curved pipe, the boundary layer separation zone is expanded by large curvature ratio, which makes the internal flow even more disordered. On the other hand, the value of I_t has improved significantly in the bend and reached the maximum value at $\theta = 90^\circ$.

Conflict of Interests

The authors declare that there is no conflict of interests regarding the publication of this paper.

References

- [1] J. Eustice, "Experiments on stream-line motion in curved pipes," *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*, vol. 85, no. 576, pp. 119–131, 1911.
- [2] W. R. Dean, "XVI. Note on the motion of fluid in a curved pipe," *Philosophical Magazine*, vol. 4, no. 2, pp. 208–238, 1927.
- [3] W. R. Dean, "LXXII. The stream-line motion of fluid in a curved pipe (Second paper)," *Philosophical Magazine and Journal of Science*, vol. 5, no. 30, pp. 673–695, 1928.
- [4] C. M. White, "Streamline flow through curved pipes," *Proceedings of the Royal Society of London. Series A*, vol. 123, no. 792, pp. 645–663, 1929.
- [5] G. I. Taylor, "The criterion for turbulence in a curved pipes," *Proceedings of the Royal Society of London, Series A*, vol. 124, no. 794, pp. 243–249, 1929.
- [6] D. J. McConlogue and R. S. Srivastava, "Motion of a fluid in a curved tube," *Proceedings of the Royal Society of London, Series A: Mathematical and Physical Sciences*, vol. 307, no. 1488, pp. 37–53, 1968.
- [7] D. Greenspan, "Secondary flow in a curved tube," *Journal of Fluid Mechanics*, vol. 57, no. 1, pp. 167–176, 1973.
- [8] W. Y. Soh and S. A. Berger, "Fully developed flow in a curved pipe of arbitrary curvature ratio," *International Journal for Numerical Methods in Fluids*, vol. 7, no. 7, pp. 733–755, 1987.
- [9] A. M. K. P. Taylor, J. H. Whitelaw, and M. Yianneskis, "Curved ducts with strong secondary motion: velocity measurements of developing laminar and turbulent flow," *Journal of Fluids Engineering*, vol. 104, no. 3, pp. 350–359, 1982.
- [10] K. Sudo, M. Sumida, and H. Hibara, "Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend," *Experiments in Fluids*, vol. 25, no. 1, pp. 42–49, 1998.
- [11] K. Sudo, M. Sumida, and H. Hibara, "Experimental investigation on turbulent flow in a square-sectioned 90-degree bend," *Experiments in Fluids*, vol. 30, no. 3, pp. 246–252, 2001.
- [12] A. Ono, N. Kimura, H. Kamide, and A. Tobita, "Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition," *Nuclear Engineering and Design*, vol. 241, no. 11, pp. 4409–4419, 2011.
- [13] L. Tan, B. Zhu, Y. Wang, S. Cao, and K. Liang, "Turbulent flow simulation using large eddy simulation combined with characteristic-based split scheme," *Computers & Fluids*, vol. 94, pp. 161–172, 2014.
- [14] M. J. Tunstall and J. K. Harvey, "On the effect of a sharp bend in a fully developed turbulent pipe-flow," *Journal of Fluid Mechanics*, vol. 34, no. 3, pp. 595–608, 1968.
- [15] J. Pruvost, J. Legrand, and P. Legentilhomme, "Numerical investigation of bend and torus flows, part I: effect of swirl motion on flow structure in U-bend," *Chemical Engineering Science*, vol. 59, no. 16, pp. 3345–3357, 2004.
- [16] F. Rütten, W. Schröder, and M. Meinke, "Large-eddy simulation of low frequency oscillations of the Dean vortices in turbulent pipe bend flows," *Physics of Fluids*, vol. 17, no. 3, p. 35107, 2005.
- [17] L. H. O. Hellström, M. B. Zlatinov, G. Cao, and A. J. Smits, "Turbulent pipe flow downstream of a 90° bend," *Journal of Fluid Mechanics*, vol. 735, article R7, 12 pages, 2013.
- [18] S. A. Berger, L. Talbot, and L. S. Yao, "Flow in curved pipes," *Annual Review of Fluid Mechanics*, vol. 15, pp. 461–512, 1983.

- [19] M. R. H. Nobari and D. Rajaei, "A numerical investigation of developing flow in concentric and eccentric curved square annuli," *Journal of Fluids Engineering*, vol. 135, no. 8, Article ID 081201, 2013.
- [20] B. T. Kuan and M. P. Schwarz, "CFD simulation of single-phase and dilute particulate turbulent flows in 90° duct bends," in *Proceedings of the ASME/JSME 4th Joint Fluids Summer Engineering Conference*, pp. 1901–1908, Honolulu, Hawaii, USA, July 2003.
- [21] M. Raisee, H. Alemi, and H. Iacovides, "Prediction of developing turbulent flow in 90°-curved ducts using linear and non-linear low-Re k - ϵ models," *International Journal for Numerical Methods in Fluids*, vol. 51, no. 12, pp. 1379–1405, 2006.
- [22] I. Di Piazza and M. Ciofalo, "Numerical prediction of turbulent flow and heat transfer in helically coiled pipes," *International Journal of Thermal Sciences*, vol. 49, no. 4, pp. 653–663, 2010.
- [23] A. S. Berrouk and D. Laurence, "Stochastic modelling of aerosol deposition for LES of 90° bend turbulent flow," *International Journal of Heat and Fluid Flow*, vol. 29, no. 4, pp. 1010–1028, 2008.
- [24] J. Zhang, A. E. Tejada-Martínez, and Q. Zhang, "Evaluation of large eddy simulation and RANS for determining hydraulic performance of disinfection systems for water treatment," *Journal of Fluids Engineering*, vol. 136, no. 12, Article ID 121102, 2014.
- [25] F. Nicoud and F. Ducros, "Subgrid-scale stress modelling based on the square of the velocity gradient tensor," *Flow, Turbulence and Combustion*, vol. 62, no. 3, pp. 183–200, 1999.
- [26] H. Ito, "Pressure losses in smooth pipe bends," *ASME Journal of Fluids Engineering*, vol. 82, no. 1, pp. 131–140, 1960.
- [27] P. L. Spedding, E. Benard, and G. M. McNally, "Fluid flow through 90 degree bends," *Developments in Chemical Engineering and Mineral Processing*, vol. 12, no. 1-2, pp. 107–128, 2004.
- [28] K. Sudo, M. Sumida, and R. Yamane, "Secondary motion of fully developed oscillatory flow in a curved pipe," *Journal of Fluid Mechanics*, vol. 237, pp. 189–208, 1992.