

Numerical and experimental investigation on turbulent flow and secondary flow in a 180° bend -- Circular Cross-section

Qing Shao^{1,2*}, Weihua Hui¹ and Futing Bao¹

¹ College of Astronautics, Northwestern Polytechnical University, Xi'an 710072, China

² Shanghai Electro-Mechanical Engineering Institute, Shanghai 201109, China

* Corresponding author's e-mail: sqbcfs@163.com

Abstract: The characteristic of turbulent flow-field in the circular-sectioned 180° bend was investigated in this study, the profiles of tangential velocity and pressure were measured by Laser-Doppler Velocimetry (LDV) and were simulated by FLUENT software. Different turbulent models are considered, including k- ϵ , RNG k- ϵ and Reynolds stress model (RSM) model. Compared with experimental measurements, the Reynolds stress model was selected to simulate turbulent flow in the bend. The results suggest that centrifugal force of fluid rotating contributes to the variation of tangential velocity and pressure in the bend. Secondary flow generating from the uneven pressure distribution in the bend was predicted by numerical simulation. Secondary flow generates near to the outer wall and shows great significance at 180° position. The secondary flow vortices moves from outer wall to inner wall gradually. This investigation indicates that turbulent flow-field characteristics in the 180° bend can be understood in details by combined experiment measurement and numerical simulation.

1. Introduction

The bends are widely used for chemical reactors, gas pipeline, heat exchangers, etc. The flow through a bend has attracted widespread attention due to its importance for transfer process in oil industry, spaceflight, environmental technology, chemical and mechanical engineering. Taylor[1] measured the laminar and turbulence flow field of the fluid in 90° bend with rectangular cross-section by LDV, he got the average velocity with different sections and the pressure distribution in curving wall at the same time and concluded that there were high pressure gradients and secondary flows which are perpendicular to the main flow direction. Some other scholars' experimental results[2-4] showed consistency with Taylor's on 90° bend flow field. The tangential velocity (which is along the main flow) of the flow in bends had been changed at 0° cross-section. The pressure on the inner wall of the bend is lower than that of the outer wall. The velocity in the shearing layer adjacent to boundary layer separation region of bend inner wall fluctuates violently and presents strong anisotropism, while the fluctuation velocity is almost at the same order of magnitude and isotropyism in the other region near to the center of the bend. For the flow field of 180° bend, Cheah[5] and Xu[6] measured the flow of 180° bend with rectangular cross-section by using LDV and Five-Hole Probe. The result deemed that the basic characteristic of flow field is similar to that of 90° bend. The main flow tangential velocity in bend also changes a lot along the radial and annular direction, which leads to the asymmetric pressure distribution in bend. The existence of axial velocity affirms the existence of secondary flow which is perpendicular to the main flow velocity in bend.



Computational fluid dynamic (CFD) is useful for prediction and optimization of various processes. With the improvement of calculation capacity and numerical algorithms, the flow in complicated geometries can be simulated.

Numerical simulation[3, 7-9] on the flow field of 90° and 180° bend further revealed the complex flow characteristics in the bend, such as secondary flow, boundary -layer separation near the inner wall, the pressure distribution near wall, deflexion of the exit-velocity in the bend, flow field pressure aberration and so on.

The flow through the bend is a 3-D flow field, and the previous researches mainly focused on the distribution of tangential velocity and pressure coefficient. It's said that the tangential velocity influences the pressure distribution and the form and change of secondary flow directly. The emergence of secondary flow originates in the velocity and the change of pressure distribution in bend, and is also affected by the structure and curvature radius. The existence of second flow makes the turbulence intensity increase, and resistance grow. Previous researches[10-11] on secondary flow in bend mainly described the shape and eddy structure by qualitative analysis, but short of quantitative analysis. Therefore, this paper investigated the flow through 180° bend by experiment and numerical simulation, explaining the basic flow mechanism of flow field.

2. EXPERIMENTAL

2.1 Experimental apparatus

The sketch of the geometry of 180° circular circular-sectioned bend is shown in Figure 1. The bend is 87 mm in inner radius (r_i) and 192 mm in outer radius (r_o). The lengths of the importing and exporting straight duct are respectively 340 mm and 840 mm.

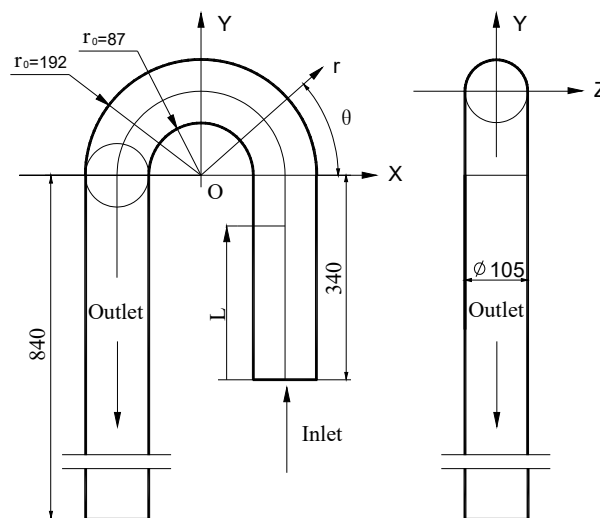


Figure 1. Sketch of the geometric parameters of the bend

2.2 Measurement method

LDV measurement

The LDV apparatus was used to measure the gas velocity in the bend. A data acquisition system (TSI 1990 A processor), was connected to a computer to convert the Doppler shifts into velocity values and produced on-line measurements of mean and fluctuating velocity. The velocity measurement range of LDV varies from 1 mm/s to 1000 m/s, with the precision of 0.1%.

The flow in the upper half part of the cross-section was taken as the main object in this analysis, since the flow fields in upper and lower half parts of the bend were symmetrical.

The non-dimensional radial distance r^* is defined as $r^* = (r - r_i) / (r_0 - r_i) - r_i / 2r_i$, where the centre of the cross-section of the bend is $r^* = 0$.

The non-dimensional axial distance $Z^* = Z/105$, where XOY plane of the bend means $Z^* = 0$.

L is defined as the flow distance from the inlet of the bend, which means the distance of 0° cross-section is 340 mm.

Operating temperature and relative humidity of the air during the experiments were separately 22°C and 45%. The gas inlet velocity was set as 10 m/s.

Pressure measurement system

The absolute pressure near the wall in the bend was measured in the experiment. In this study, 16 positions measured in the bend are the positions near the inner and outer wall in 7 cross-sections

3. NUMERICAL PROCEDURE

3.1 Turbulent airflow model

Turbulent flows are simulated using methods of the resolution of Reynolds time-averaged. Navier–Stokes equations. A three dimensional, stable, incompressible, turbulent and continuous fluid was assumed in this study. The mass conservation equation and momentum balance equation are shown as follows:

$$\frac{\partial}{\partial x_i}(\rho \bar{u}_i) = 0 \quad (1)$$

$$\frac{\partial}{\partial x_i}(\rho \bar{u}_i \bar{u}_j) = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial}{\partial x_j}(\rho \bar{u}_i' \bar{u}_j') + \rho g_i \quad (2)$$

where x_i , u_i , ρ , g_i and μ are the spatial co-ordinate, the velocity along i -direction, the fluid density, the mean pressure, the gravity component along i -direction and dynamic viscosity, respectively.

In equation (2), the Reynolds stress $\rho \bar{u}_i' \bar{u}_j'$ could be taken by Boussinesq's assumption which is employed in the k - ε model and RNG k - ε models

$$\bar{u}_i' \bar{u}_j' = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k \quad (3)$$

where μ_t is the turbulent viscosity and $k = 0.5 \bar{u}_i' \bar{u}_i'$ is the turbulent kinetic energy.

$$\mu_t = C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

where ε is the turbulence dissipation rate.

k and ε can be calculated by the two following supplementary equations[12]:

k -equation:

$$\frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \mu_t \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \frac{\partial u_i}{\partial x_j} + G_b - \rho \varepsilon \quad (5)$$

ε -equation:

$$\frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon 1} \mu_t \frac{\varepsilon}{k} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \frac{\partial u_i}{\partial x_j} - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad (6)$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_μ , σ_k , σ_ε are shown in table 1.

In the RNG model, the turbulent kinetic energy and its dissipation should be obtained through k -equation (5) and ε equation (6). The effective viscosity $\nu(I)$ is given by the RNG theory:

$$\nu(l) = \nu_{mol} \left[1 + \frac{2A\varepsilon}{4\nu_{mol}^3} (l^4 - l_d^4) \right]^{1/3}, \quad l \geq l_d \quad (7)$$

In the RSM model, the Reynolds stresses are computed individually by solving transport equations. The triple order velocity correlations that appear as unknowns in these equations are then modelled as follows:

$$\overline{u_k \frac{\partial}{\partial x_k} (\rho \overline{u_i' u_j'})} = \frac{\partial}{\partial x_k} \left(\frac{\nu_t}{\sigma_k} \frac{\partial \overline{u_i' u_j'}}{\partial x_k} \right) + P_{ij} + \phi_{ij} - \varepsilon_{ij} + R_{ij} \quad (8)$$

where the stress production rate P_{ij} is given by:

$$P_{ij} = -\overline{\rho u_j' u_k'} \frac{\partial u_j}{\partial x_k} - \overline{u_j' u_k'} \frac{\partial u_i}{\partial x_k} \quad (9)$$

$$\phi_{ij} = -C_3 \frac{\varepsilon}{k} \left(\overline{u_i' u_j'} - \frac{2}{3} \delta_{ij} k \right) - C_4 \left(P_{ij} - \frac{2}{3} \delta_{ij} P \right) \quad (10)$$

$$\varepsilon_{ij} = \frac{2}{3} \delta_{ij} \varepsilon \quad (11)$$

$$R_{ij} = -2\Omega_k \left(\overline{u_i' u_j'} \varepsilon_{ikm} + \overline{u_i' u_m'} \varepsilon_{jkm} \right) \quad (12)$$

The constant values present in Table 1. The fluid density ρ and viscosity μ were used as constants.

Table 1. Constant value in the turbulent flow models.

MODEL	k - ε			RNG k - ε			RSM		
Item	C_μ	σ_k	σ_ε	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	α	C_μ	C_3	C_4
Value	0.09	1.0	1.3	1.44	1.92	1.39	0.0845	1.8	0.6

3.2 Numerical procedure

3.2.1 Grid meshes

The grid independency was tested using computational grids between 50,000 and 200,000 cells. The computational grid of near 150,000 cells with cross-section of 640 cells expressed good performance, which meant this grid made well- predicted.

3.2.2 Boundary conditions

The turbulent air flow was assumed as a fully developed turbulent flow. The inlet velocity was used as inlet boundary. The outlet boundary condition was declared as a pressure outlet boundary. All walls were set up as a non-slip wall boundary condition. The fluid property of dynamic viscosity $\mu = 1.789 \times 10^{-5}$ kg s/m and density $\rho = 1.225$ kg/m³ at T = 288 K were used in the simulations.

3.3 Validation of CFD models

All the models, the standard k - ε model, the RNG k - ε model and the Reynolds stress model, available in Fluent software, have been extensively used. All those studies show that various numerical modeling approaches can be retained for bend flows. Results in curved pipes are difficult to extend to that of swirling flows in bends, where hydrodynamic conditions are greatly different in the swirl structures.

The numerical model's efficiency is evaluated in this section by comparing the results of these three simulations with the experimental results, to select the most efficient model for the case of the study.

Predictions obtained by these three different turbulent models are shown in Figure 2. All velocities are provided as velocities non-dimensionalized by inlet velocity (Figures 2-4).

As is shown in Figure 2, all turbulence models presented almost similar predictions but employed lower predictions than the experiment measurement. At $\theta=0^\circ$, the calculated tangential velocities of three models showed great agreement with experimental results in the bend. At $\theta=120^\circ$, $\theta=150^\circ$, $\theta=180^\circ$, predictions from RSM were more precise than that from other simulation models, due to the evident radial pressure gradient and the anisotropy. The transport equations for RSM are solved by the Reynolds stress tensor; while in the other models, the Boussinesq hypothesis is assumed to relate the Reynolds stresses to the mean velocity gradients. Therefore, the Reynolds stress model was used to simulate the turbulent flow.

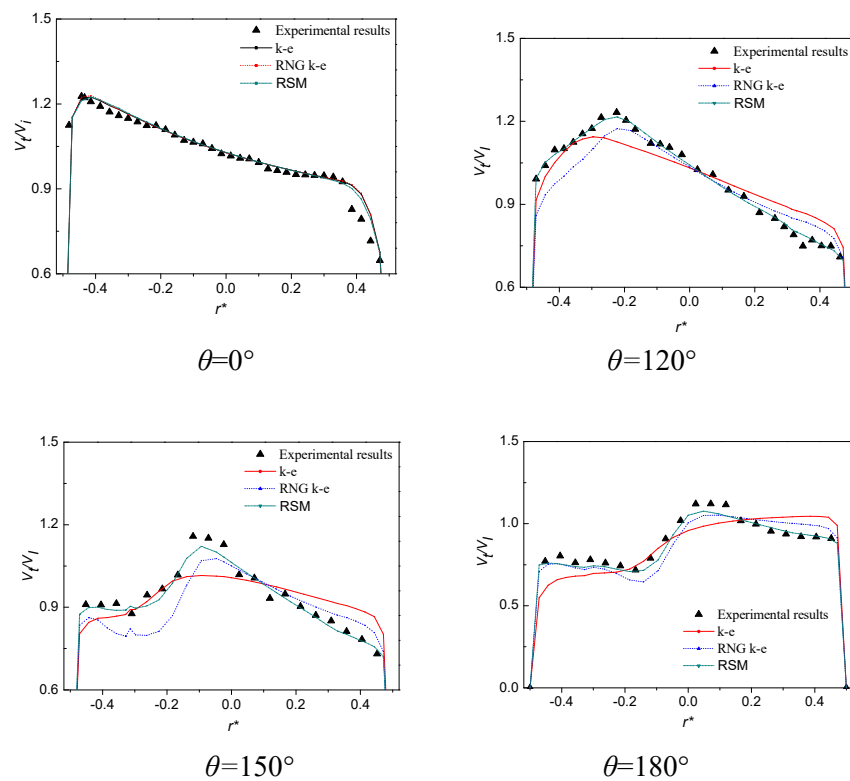


Figure 2. Results of different numerical models (mean tangential velocity profile along r^*)

4. RESULTS AND DISCUSSION

4.1 Tangential velocity

Tangential velocity is the most important parameter in the bend. The flow which goes from the straight duct to the bend restricted by the bend wall is conducted to the swirl flow, which generates the centrifugal force in direction to the outer wall of the bend. This force makes the radial pressures uneven and forms the pressure gradient. Therefore, this centrifugal force is equal to this radial pressure gradient:

$$\frac{u_t^2}{r} = \frac{1}{\rho} \frac{\partial p}{\partial r} \quad (13)$$

The pressure gradient in direction of radius induces the uneven tangential velocity distribution. The influence of the radial pressure gradient, the boundary layer effect near the inner wall and wall

friction make the tangential velocity of the gas flow near the inner wall decrease rapidly.

The non-dimensionalized tangential velocity profiles at the 6 typical positions ($\theta=0^\circ$, 30° , 60° , 90° , 120° , 150° and 180°) are compared with the experimental measurements in Figure 3. The experimental measurements are almost well-predicted. The change of the tangential velocity starts at $\theta=0^\circ$. The centrifugal effect of the bend shifts the tangential velocity to maximum toward the inner wall and minimum toward the outer wall at around $\theta=60^\circ$. Then the tangential velocity decreases along the inner wall and increases along the outer wall from $\theta=60^\circ$ to $\theta=180^\circ$ gradually.

The direction of the flow in the bend consists of the main flow (tangential direction) and secondary flow (perpendicular to the main flow). Compared to the straight flow, the flow pattern in the bend is more complicated. The turbulent intensity and the frictional drag in the bend are higher than that in the straight pipe.

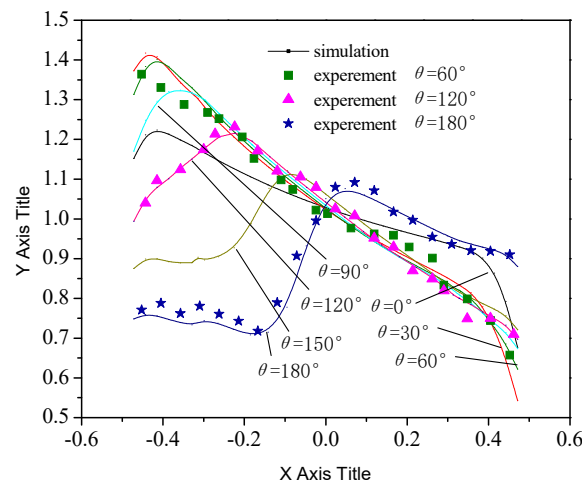


Figure3. Distribution of tangential velocity in the midplane ($Z=0$)

4.2 Pressure

Non- dimensionalized pressure coefficient C_p is defined to study the static pressure in the bend:

$$C_p = (P_s - P_{ir}) / \left(\frac{1}{2} \rho V_i^2 \right) \quad (14)$$

It is seen in Figure 4 that the pressure in the inner wall is higher than that in the outer wall due to the centrifugal force. The pressure near the inner wall decreases while that near the outer wall increases gradually from section $\theta=0^\circ$ to section 30° . And next, the pressure near the inner wall increases while that near the outer wall decreases from section $\theta=30^\circ$ to section 180° . At the section of $\theta=30^\circ$, the radial pressure gradient reaches to the maximum. Therefore, the flow near inner wall is convergent flow while that near outer wall is diffusion flow from $\theta=0^\circ$ to 30° . The flow near inner wall becomes diffusion flow while that near outer wall becomes convergent flow from $\theta=30^\circ$ to 180° .

Figure 5 shows the pressure coefficient along the flow direction in the inner and outer wall. From the curve of the pressure distribution, the energy loss in the bend mainly happens near the inner wall, especially during $\theta=30^\circ$ to 90° , which is due to the separation phenomenon of the flow and the variation of the boundary layer among the flow process. The pressure in the outer wall is always higher than that in the inner wall. The asymmetrical distribution of pressure in the bend demonstrates the balance of pressure and centrifugal force. In the meanwhile, centrifugal force is proved to somehow reflected to the distribution of the tangential velocity (Figure 3).

Figure 6 is the whole pressure distribution in the bend, which is consistent with the result in Figure 4 and Figure 5.

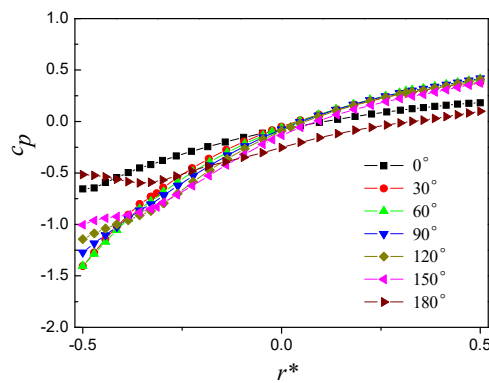
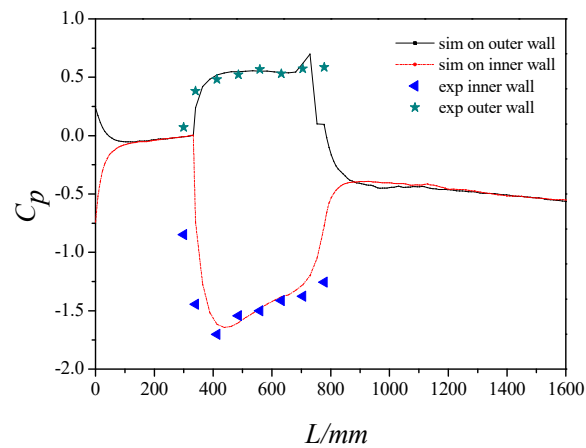
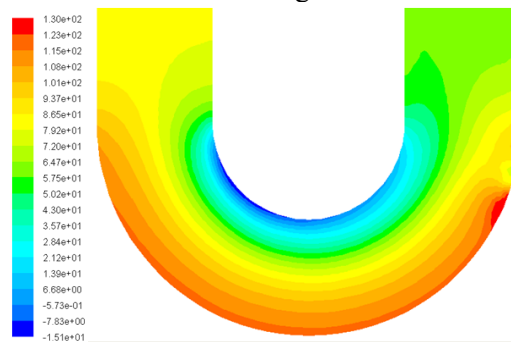
Figure 4. Pressure distribution along direction of r^* 

Figure 5. Pressure distribution along the inner wall and outer wall

Figure 6. The distribution of C_p in the bend

4.3 Secondary flow

4.3.1 Formation of secondary flow

With the equation(13), the centrifugal force has to be balanced with the radial pressure gradient. But the tangential velocities in different positions of the cross-sections are different and the centrifugal forces are not equal to each other from the above analysis in section 4.1.

At a given cross-section, the tangential velocities near the top (S) or bottom (U) points are lower than that in the centre due to the effect of the frictional resistance. The radial pressure gradient correspondingly exists along the Q-T direction. The pressure at point T (outer wall) is higher than that at point Q (inner wall), also higher than that at point S and U. This pressure difference makes the axial

and radial velocities perpendicular to the tangential direction, which generates the secondary flow. Two symmetrical eddies are observed in Figure 7. Therefore, the radial pressure gradient could be considered as the root of the secondary flow.

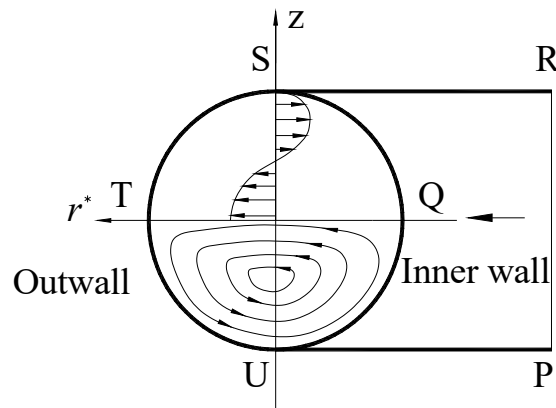
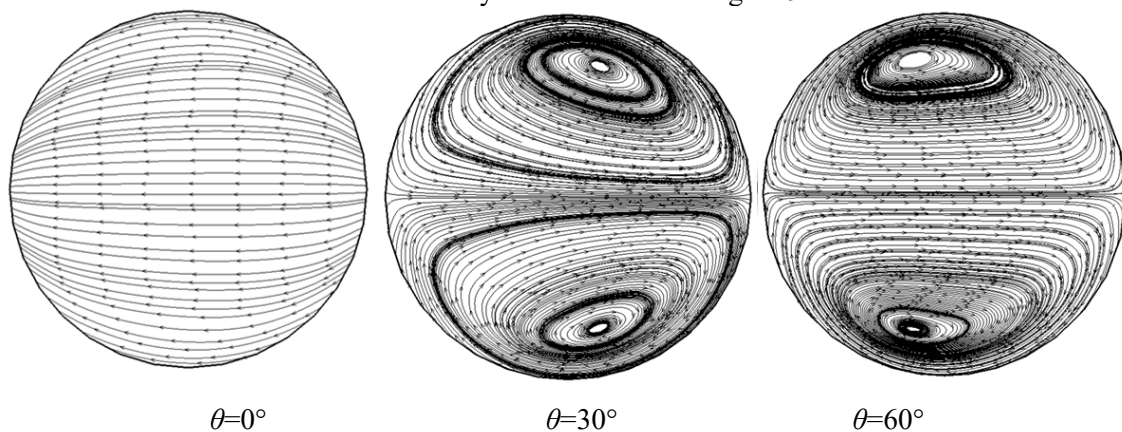


Figure 7. Secondary flow in the bend

4.3.2 Track of secondary vortices

The formation and development of the secondary flow along the bend (cross-section from $\theta=0^\circ$ to 180°) can be seen in Figure 8. The flow starts being changed into swirl flow from $\theta=0^\circ$. At section $\theta=30^\circ$, The existence of the axial and the radial velocities due to the radial pressure gradient results in two vortices ($r^*=0.145$, $Z^*=\pm 0.33$) at the right corner. At section $\theta=60^\circ$, the vortices moves towards to the inner wall located at ($r^*=-0.10$, $Z^*=\pm 0.39$). The vortices is at ($r^*=-0.27$, $Z^*=\pm 0.28$) when at section of $\theta=90^\circ$. Passed the section $\theta=90^\circ$, the vortices moves not only towards to the inner wall but also to the centre ($Z=0$). When the flow goes through the exporting straight pipe, the vortices move away from the inner wall.

The track of the vortices of the secondary flow is shown in Figure 9.



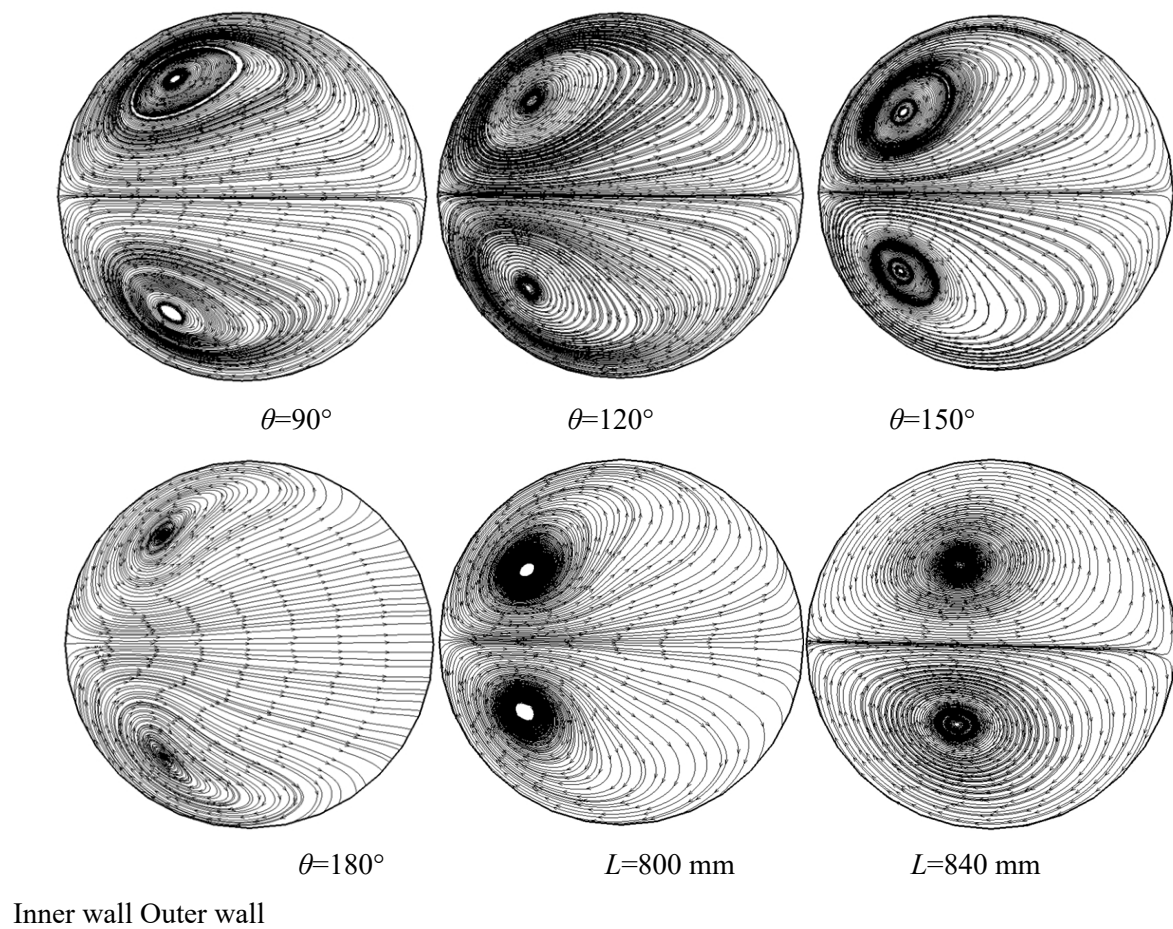


Figure 8. Velocity vector profiles at the top of different longitudinal cross-sections

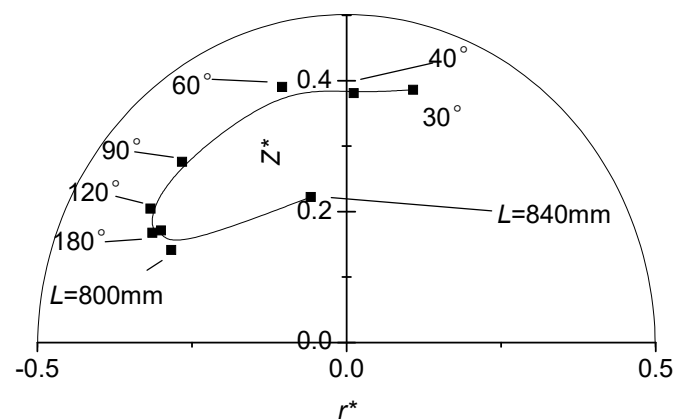


Figure 9. Movement track of secondary vortices

4.3.3 Intensity of secondary flow

The relationship between moment of momentum (based on the vortex center at different cross-sections) and initial moment of momentum is investigated to describe the variation of the intensity of secondary flow. The intensity of secondary eddy K is defined by:

$$K = \frac{\iint_{\theta} \vec{V}_{rz} \times \vec{r}' dA}{\iint_{ab} \vec{V}_i \times \vec{r} dA} \quad (15)$$

where $\vec{V}_{rz}$ is the rotation velocity vector of secondary eddy, $\vec{r}'$ is the distance vector based on the vortex center at a certain deflection of the bend, is the distance vector to the vortex center at the bend inlet. The intensity of secondary eddy (Figure 10) increases sharply during $\theta = 0^\circ$ - 90° and rises gently from $\theta = 90^\circ$ to 180° , then gets the maximum value that occupies 25% of initial momentum at the inlet. However, the intensity of secondary flow attenuates after going through the exporting straight pipe.

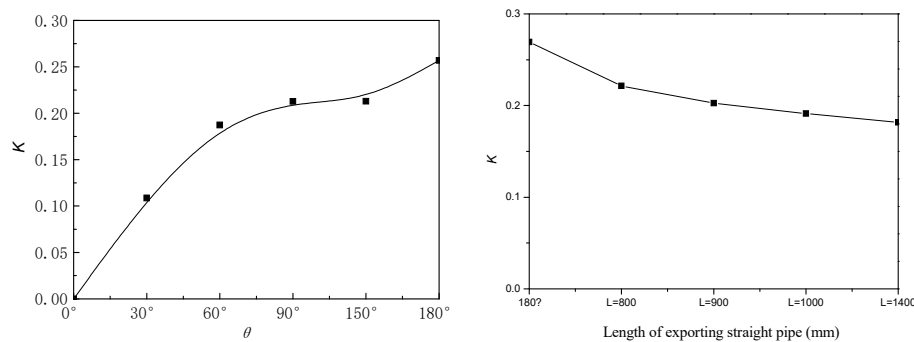


Figure 10. Intensity of secondary flow at different cross-sections

5. CONCLUSIONS

The turbulent flow in the circular-sectioned 180° bend is a complex of the mainstream tangential velocity and secondary flow, and presents complicated three-dimensional flow behavior.

To get a good prediction of the flow in the bend, three common turbulent models, including $k-\varepsilon$, RNG $k-\varepsilon$ and Reynolds stress model (RSM) model, were compared by experimental measurements. The testing results prove that RSM is of highest prediction precision on the turbulent flow that expresses strong anisotropy in the bend among these three models.

The measurement and numeric simulation showed that: near the inner wall, the tangential velocity increase while the pressure decreases from 0° to 60° and then the velocity is reduced while the pressure grows higher from 60° to 180° ; on the contrary, near the outer wall, the tangential velocity drops while the pressure rises from 0° to 60° , and then the velocity goes down while the pressure goes up from 60° to 180° . The tangential velocity near the inner wall is always bigger than that near the outer wall while the pressure near the outer wall is higher than that near the inner wall.

The effect of the bend curvature and the wall frictional resistance results in the change of the tangential velocity and the uneven distribution of the pressure in the cross-section, engendering the secondary flow. As the fluid going, the center of the secondary flow moves from outer wall to inner wall gradually, meanwhile the effected area and the intensity increase in the 180° bend. At the section of 180° position, the intensity of secondary flow reaches to the strongest. After the fluid flows into the straight duct, the centre of the secondary flow goes from inner wall back to outer wall, simultaneously the area and intensity of the secondary flow are rapidly weakened, till the secondary flow decays and vanishes.

REFERENCES

- [1] Taylor, A., Whitelaw, J., Yianneskis, M. (1982) Curved ducts with strong secondary motion-Velocity measurements of developing laminar and turbulent flow. Transactions Journal of Fluids Engineering, 104: 350-359.

- [2] Jiang, H., Lu, L., Sun, K. (2011) Experimental study and numerical investigation of particle penetration and deposition in 90° bent ventilation ducts. *Building and Environment*, 46: 2195-2202.
- [3] Kuan, B., Yang, W., Schwarz, M. (2007) Dilute gas–solid two-phase flows in a curved 90° duct bend: CFD simulation with experimental validation. *Chemical engineering science*, 62: 2068-2088.
- [4] Abhari, M.N., Ghodsian, M., Vaghefi, M., Panahpur, N. (2010) Experimental and numerical simulation of flow in a 90° bend. *Flow Measurement and Instrumentation*, 21: 292-298.
- [5] Cheah, S., Iacovides, H., Jackson, D., Ji, H., Launder, B. (1994) LDA investigation of the flow development through rotating U-ducts. *Journal of turbomachinery*, 118: 590-596.
- [6] Xu, J., Du, C., Wang, T., Wei, Y., Shi, M. (2010) Experimental measurement of flow field in 180° curved duct with rectangular cross2section by LDV. *Chinese Journal of Experiments in Fluid Mechanics*, 24: 36-41.
- [7] Launder, B.E., Spalding, D. (1974) The numerical computation of turbulent flows. *Computer methods in applied mechanics and engineering*, 3: 269-289.
- [8] Luo, J., Lakshminarayana, B. (1997) Prediction of strongly curved turbulent duct flows with Reynolds stress model. *AIAA journal*, 35: 91-98.
- [9] Pruvost, J., Legrand, J., Legentilhomme, P. (2004) Numerical investigation of bend and torus flows, part I: effect of swirl motion on flow structure in U-bend. *Chemical engineering science*, 59: 3345-3357.
- [10] Fan, H., Li, X., He, Z. (2002) Numerical simulation of the secondary flow in a curved duct of square-shaped cross-section. *Journal of Engineering for Thermal Energy and Power*, 17: 510-513.
- [11] Breuer, M., Baytekin, H., Matida, E. (2006) Prediction of aerosol deposition in 90° bends using LES and an efficient Lagrangian tracking method. *Journal of aerosol science*, 37: 1407-1428.
- [12] Yang, W., Kuan, B. (2006) Experimental investigation of dilute turbulent particulate flow inside a curved 90 bend. *Chemical engineering science*, 61: 3593-3601