Comparative Analysis of the Flow Field in the Lower Bend at Different Rotation Angles

Zhaoliang Liu¹, Jinlong Wang² and Chengdong Yuan²*

¹ Shandong university of science and technology, Shandong, China
² School of Weifang university, Shandong, China

*Corresponding author e-mail: wjl006908@163.com

Abstract. In order to obtain the influence of the distribution of the flow field in the lower bend at different rotation angles on the fluid flow, Fluent software was used to conduct numerical simulation of the flow field in the bend at different rotation angles, and the influence rule of the shape difference of the secondary flow in the lower bend at different rotation angles was simulated. The simulation results show that the 90 ° rotation Angle bend, its internal gas phase flow field better than other simulation Angle, secondary flow phenomena delay happened, reduce reflux area, syphon is helpful to improve the flow state; With the change of the rotation Angle, the tangential velocity of the outlet section of the bend increases continuously, which is conducive to the smooth gas-solid phase separation, but the pressure drop of the system also increases, resulting in the energy loss of the system. As a result, the Angle of 90 ° canary after rotation Angle for the simulation of the optimal point of view, for later engineering application provides data to support and help.

1. Introduction

In petroleum, chemical industry, machinery, power, water conservancy and other engineering applications, it is often designed to different rotation Angle of the bend flow problem. Taylor[1] made a great contribution in the early research period. He used LDV technology to make a detailed experimental study on the flow rate and pressure distribution of laminar and turbulent flows in horizontal pipe bends, and clearly proposed that there was a large pressure gradient and a secondary flow perpendicular to the flow direction in the pipe bend. Humphrey[2] also carried out experimental research on turbulent flows in different cross-sections in the horizontal bend, obtained flow field characteristics and pressure distribution data, and realized that the curvature of the bend had a certain impact on its internal turbulent flows. Hong Shang et al.[3] application of CTA hot wire anemometer and five hole probe to measure the circular cross section curvature at 0.87 level of 90 ° bend the inside of the turbulent flow, and the bent pipe under different Reynolds number is given different cross section parameters such as velocity, turbulent kinetic energy, total pressure distribution. In terms of theoretical research on curved pipes, Jue Ding et al.[4] studied the effects of different inlet directions on the structure and flow characteristics of the flow field in a horizontal bend by using the RNG - turbulence model. Hongming Fan et al.[5] used Largeeddysimu2lation (LES) method to simulate the fluid flow in a horizontal bend in three-dimensional space, which deepened the understanding and analysis of the flow in the bend. The basic equation of the large vortex model[6,7] is a numerical simulation and prediction method based on the cascade decay theory of statistics and vortices. However, the above research objects mainly focus on the horizontal bend. Although the influence of
different flow directions on the flow field in the bend has been studied in the literature, the influence of the structure change of the rotation Angle of the bend on the flow field has not been taken into account.

Many bend in the engineering application is a rotation Angle, its structure and horizontal bend, there exist certain differences even in the same flow conditions, the internal flow field and horizontal bending also exist bigger difference, the exit flow field often exist velocity deflection and stress distortion, fluid energy loss not only, and can lead to erosion wear and bend particles in the wall of the deposition, affecting the normal operation of the device. Therefore, it is necessary to study the gas-phase flow field of such a bend. Fluent software was used to conduct numerical simulation of the flow field in the bend at different rotation angles, and the influence rules of secondary flow and jet flow in the bend at different rotation angles on the fluid flow in the pipe were mainly investigated, so as to provide references for engineering design.

2. Geometric model and boundary conditions

2.1. The geometric model

As shown in Figure 1, the elbow is divided into horizontal section, bending section and outlet section (its geometric size is shown in Figure 1). As the rotation Angle of the bend, represents the curvature of the bend, $c$ is at the near center of the bend, $d$ is at the far center of the bend, and the overall size of the bend is shown in the following table 1.

| Table 1 geometric dimensions of elbow |
|-------------------------------------|
| size | a(mm) | b(mm) | $\theta$ (°) |
| Data | 70 | 60 | variable |

Figure 1. geometric sketch of the elbow

2.2. The boundary conditions

(1) Inlet boundary conditions for fluid to flow uniformly into the bend, its inlet velocity $v = 35$ m/s, and inlet turbulence intensity $I = 4.21\%$.

(2) The exit conditions were treated according to the fully developed pipe flow conditions, that is, the axial gradient of all variables at the exit section was zero.

(3) Wall condition the wall is treated by non-slip boundary condition and the standard wall function method is used in the vicinity of the wall.
3. Simulation results and analysis

3.1. Simulation and analysis of pressure distribution of lower elbow with different rotation angles

As shown in Figure 2. Syphon in order to study the different pressure distribution of flow field simulation scheme take 60° bend 120°, 90° bend but bend data simulation and analysis, through to the three different curved pipe pressure field simulation, analysis of the stress field in the bend structure distribution.

According to the simulation analysis results, it can be seen that under the three simulated environments, the pressure at the proximal center is lower than that at the distal center, which is consistent with the existing research[8]. When the rotation Angle 60°, curved pipe pressure injection close at the heart of change, in the region of the exports, the pressure drop caused by the corner big, make close to the inside of the inlet pressure is greater than near the inner of the exit. Because of the difference of pressure distribution, there will be a secondary flow at the outlet of the elbow. Syphon when the rotation Angle 90°, near the heart without injection pressure changes, in export area and pressure drop phenomenon, no secondary flow phenomena happened in bend the exit; When the rotation Angle 120°, bending tube near the heart jet is 60°more obvious change, at the exit, a greater pressure drop area, secondary flow phenomenon is more obvious than when 60°. Comprehensive the
above analysis, in the numerical simulation of stress field, is superior to 90° rotation Angle 60° and 120°.

3.2. Velocity distribution and simulation analysis of lower elbow with different rotation angles

As shown in Figure 3, Syphon to study the flow field velocity distribution, simulation scheme take 60°, 90° and 120° bend data simulation and analysis, through the simulation of velocity field in three different bend tube, analysis the distribution of velocity field within the bend structure.

According to the simulation analysis results, it can be seen that there are certain dead zones in the inner wall of the elbow under the three simulated environments. Syphon when the rotation Angle 60°, the central area of velocity field is uniform, but in recent heart point near the exit jet flow phenomenon exists, will affect the use of the bent pipe; Syphon when the rotation Angle 90°, the central area of velocity field is uniform, near the heart without bending tube jet flow phenomenon, near the heart flow velocity is higher than far heart; Syphon when the rotation Angle 120°, the central region is relatively homogeneous at the entrance, but close at the heart, jet flow phenomenon is more serious than the 60°,

Figure 3. (d), (e), (f) respectively 60°, 90° and 120° bend velocity field distribution
so much so that the exit velocity of stratification. Serious impact on the use of elbow. Based on the analysis of the above, is superior to 90° rotation Angle 60° and 120°.

4. The distribution of flow velocity and pressure field is summarized
From the above analysis, it can be seen that the pressure field and velocity field under different rotation angles all present regular distribution. When the rotation Angle is too large or too small, jet flow, secondary flow and other phenomena appear, which is not conducive to the flow of fluid in the elbow and affects the use of the elbow. In the experiment selected three angles, the most suitable 90° bend. Reflected in the vicinity of 90°, the distribution of pressure field and velocity field showed a trend of kind of normal distribution. Therefore, in the actual production application, according to the actual needs, to choose the right Angle of rotation to produce the elbow.

5. Conclusion
(1) In the pressure field distribution at different rotation angles, the pressure at the proximal center is smaller than that at the distal center. Therefore, the strength at the distal center should be strengthened in practical use. Measures such as inner wall thickness at the distal center and so on can be adopted.

(2) Different rotation Angle of stress field distribution, too big or too small will produce secondary flow, affect the bent pipe is used, according to the simulation data, when take 90° as the best rotation Angle.

(3) In the velocity field distribution with different rotation angles, there is a certain dead zone in the inner wall, which is caused by a variety of factors. Measures such as increasing the inlet velocity and reducing the inlet diameter can be adopted to solve this problem.

(4) Different rotation Angle velocity field distribution, too big or too small will produce jet flow phenomenon, affect the use of the bent pipe, according to the simulation data, when take 90° as the best rotation Angle.

References
[1] Taylor A M K P, Whitelaw J H, Yianneskis M. Curved ducts with secondary motion: velocity measure of developing lam and turbulent flow. Journal of Fluids Eng.1982,104(2):350-359.
[2] Humphrey J A C. Turbulent flow in a square duct with strong curvature Fluid Mech.1981,103:443-463.
[3] Hong Shang, Qingli Kong, Shangjin Wang. 90° bend tube with circular cross section of three-dimensional turbulent flow experimental research. Journal of aerospace power. 1994, 9(3):263-266.
[4] Jue Ding, Peifen Weng. 90° bend pipe flow of theoretical models and numerical studies of flow characteristics. Journal of computers. 2004, 21(3):314-321.
[5] Hongming Fan, Zhongyi He, Xiaohua Wang. Large eddy simulation of flow in curved pipe sections. Research and progress in hydrodynamics. 2001, 16(1):78-83.
[6] Zhaoshun Zhang, Guixiang Cui, Chunxiao Xu. Turbulence theory and simulation. Beijing tsinghua university press. 2005:236.
[7] Griffiths W D, Boysan F. Computational fluid dynamics (CFD) and empirical modeling of the performance of a number of cyclone samplers. Aerosol Science. 1996, 27(2):281-304.
[8] Ming Li, Jing Gong. Mathematical model of severe slug flow in Marine riser system [J]. Oil and gas storage and transportation. 2013, 32(5):462-46.