

Flow Field Analysis and Energy Saving in Bend

Mingyue Sun

School of North China Electricity Power University, Baoding 071000, China

978573307@qq.com

Keywords: bend, curvature radius, local loss, Numerical Simulation.

Abstract. Bends are widely used in piping systems. However, under the action of various factors, the flow field at the elbow is relatively complex, which may result in large local loss and increase the overall flow loss. In the general trend of promoting energy saving and emission reduction, the academic circles and engineering circles in various countries have paid great attention to analyzing the flow field inside the elbow and stabilizing it by installing the guide plate. The purpose of this paper is to review the present research situation of retrofitting baffles in China, and to explore the research methods that can summarize the universal laws.

1. Introduction

Pipeline system is widely used in the industrial field. The pipe is a common part of the pipeline system. Due to pressure pulsation, cavitation, water hammer and many other factors in the flow of the elbow, the flow field presents a very complex flow characteristics. Many bends in the projects have serious wear problems and fouling problems, uneven distribution of flow field is one of the main reasons. Therefore, it is necessary to study how to improve the flow characteristics within the elbow. Due to the complexity of the flow field inside the elbow, it is difficult to obtain the details of its internal use only by the traditional experimental method. Many of the characteristics of the elbow is not very understanding, so it is difficult to choose the appropriate elbow in particular situations. With the continuous development of computational fluid dynamics, CFD software Fluent can be used to simulate the flow of liquid inside the elbow, analyze the flow field inside the elbow and find out its variation.

2. Analysis of internal flow field of elbow

At present, the analysis of the internal flow field of the elbow has been relatively mature. The flow state at the elbow is more intense, and the vortex loss and the secondary flow loss are due to the pressure difference.

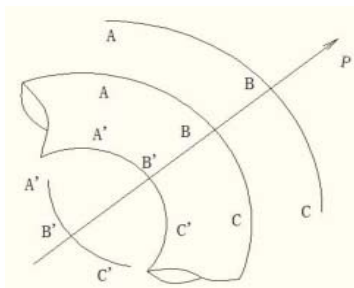


Fig. 1 Vortex loss

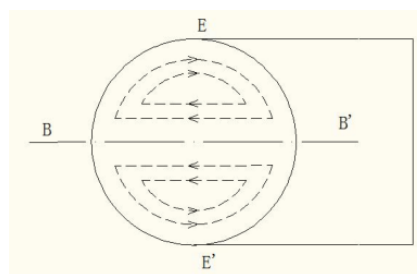


Fig. 2 Secondary flow loss

2.1 Vortex loss

Fluid flow through the curved pipe, the fluid particle must be subject to the role of centrifugal inertial force, in order to balance the centrifugal inertial force, bending the lateral pressure of the pipeline increased, the inner pressure decreased. In the case of incompressible homogeneous fluid, where the change of bit energy can be neglected, according to the Bernoulli equation, the sum of the pressure energy and the kinetic energy is constant along the streamline in the short distance. Therefore, the velocity must be reduced. On the contrary, the pressure is low, the speed must increase.

From the boundary layer theory, it can be seen that when the fluid flows through the curved wall, the boundary layer separates and forms a vortex in the deceleration zone. As shown in Fig. 1, the AB and B'C', In the AB and B'C' area will produce vortex, the formation of vortex loss, the size of the whirlpool depends on the degree of bending of the tube, the tube bent more powerful, due to the greater the energy loss caused by the vortex.

2.2 Secondary flow loss

The secondary flow occurs in a flow perpendicular to the flow plane. The pressure on the outside of the elbow is higher than the pressure on the inside, as shown in Fig. 2, and the pressure at B is higher than the pressure at B'. On the other hand, the upper and lower sides of the elbow (ie, E, E') near the wall due to low flow rate, centrifugal inertia force is small, so the pressure is also small, so that the formation of a pipe along the wall from the outside. The pressure inside the drop, that is:

$$p_B > p_E > p_{B'}, p_B > p_E > p_{B'}$$

Resulting in the flow of fluid along the wall from the outside to the inside. Due to the continuity and the action of the centrifugal inertial force, the fluid at B' flows from inside to outside along the BB' line, and two circulation flows in the radial plane, namely, secondary flow. The two secondary flows are superimposed with the main flow so that the fluid mass passing through the elbow is subjected to helical motion, that is, the energy loss through the elbow fluid.

3. Research on Energy Saving in Bend

3.1 Model establishment

The geometric model of the 90° circular elbow is shown in Fig. 3.

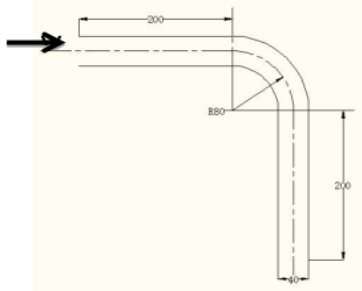


Fig. 3 Geometric diagram of the elbow

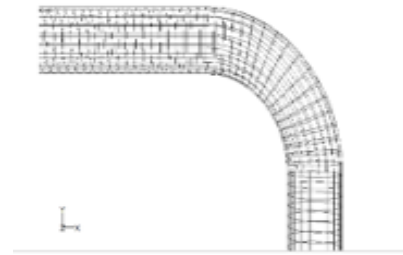


Fig. 4 The grid model of the elbow calculation domain

By controlling the transient continuity equation and the Navier-Stokes equation, the governing equations of steady and incompressible fluid flow in Cartesian coordinate system are obtained

Continuity Equation:

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

Momentum equation:

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho \bar{u}_i' \bar{u}_j' \right] - \rho g \delta_{i2} \quad (2)$$

Where \bar{u}_i , \bar{p} are the average velocity and average pressure of the fluid point, \bar{u}_i' is the velocity of the fluid pulsation, and S_i is the fluid microfacies,

$$S_i = -\rho g \delta_{i2} \quad (3)$$

Reynolds stress:

$$\tau_{ij} = -\rho \bar{u}_i' \bar{u}_j' = \mu_t \left[\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right] - \frac{2}{3} \left[\rho k + \mu_t \frac{\partial \bar{u}_i}{\partial x_i} \right] \delta_{ij} \quad (4)$$

Where μ_t is the turbulent viscosity, δ_{ij} is the Kronecker sign, and $k = \frac{\bar{u}_i' \bar{u}_i'}{2}$ is the turbulent kinetic energy, and the Boussinesq is assumed to introduce the k-ε equation using the two-equation model.

3.2 Original parameters

The k-ε model of the fluent software is used to simulate the case without the addition of the baffle. The resulting results are as follows:

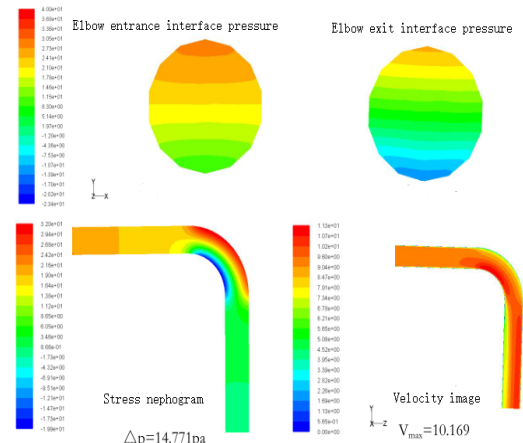


Fig.5 Numerical Simulation of No Baffles

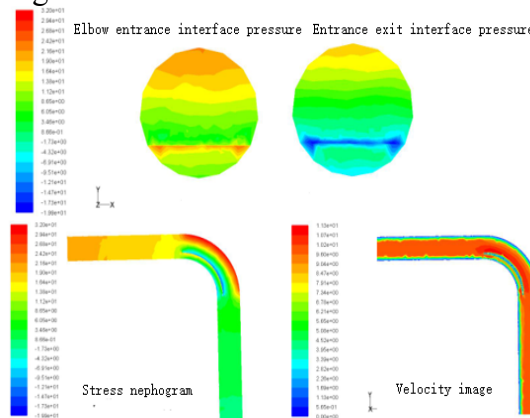


Fig. 6 Numerical simulation of the 90 ° baffle was carried out at $R = 70$

It can be seen that there is a noticeable flow unevenness at the elbow, and the elbow exit interface has a significant pressure loss compared to the inlet interface due to the instability of the flow. When the baffle without baffle, the fluid in the inertia and centrifugal force under the joint action in the vicinity of the pipe wall to form a separation zone, particle phase and gas phase separation, the pipeline cross-section will produce secondary flow.

3.3 Install the 90 ° deflector at $R = 70$

On this basis, consider adding appropriate and deflector to stabilize the flow. The simulated results of the 90 ° deflector at $R = 70$ are shown in the figure:

The pressure field distribution at the initial velocity of 10 m / s and the temperature $T = 300$ K is shown in Fig. 6. It can be seen from Fig. 6 that the pressure at the outlet of the elbow increases obviously when the baffle is not installed. If the speed is large, the increase of the speed will not cause a significant increase in the pressure inside the elbow. The decrease in the internal pressure of the elbow is a change in the gradient due to the loss of energy in the process of movement and the reduction of the outlet pressure. The pressure is small in the vicinity of the inner wall of the bend and the pressure at the outer wall of the pipe Large, this is because the fluid in the course of the movement of centrifugal force, and through a large number of simulations can be seen that the pressure increases with the speed increases.

The velocity field distribution upstream of the elbow is clearly concentric, which is due to the fact that the flow field is not affected by the bending of the elbow and the secondary flow of the liquid. When $\theta = 90^\circ$, the fluid into the curved section, the velocity field is biased towards the inner wall of the elbow and the speed increases. It can be seen that the maximum velocity of the velocity is near the inner wall surface, and the secondary flow is formed on the inner wall surface of the fluid with a small velocity at the outer wall due to the presence of the velocity difference. This is disadvantageous for long distance liquid transport.

4. Conclusion

1) At the entrance of the bend of the elbow, the main velocity near the inner wall of the elbow is increased and the secondary flow moves from the outer wall to the inner wall. Due to the presence of the secondary flow, the whole bending section exhibits a large velocity of the outer wall surface pressure, and the inner wall surface exhibits a small pressure and a small speed.

2) The existence of energy loss of the elbow, resulting in the elbow straight line section of the pressure value than the upstream straight section of the pressure value is small.

3) Using the RNG k- ϵ turbulence model in the Fluent software and the near-wall function method, the numerical simulation of the internal fluid of the elbow can simulate the secondary flow in the elbow well. In the future simulation work can also try other turbulence model, so that the elbow inside the secondary flow simulation more accurate.

4) Using the RNGk- ϵ turbulence model in FLUENT, the numerical results are in good agreement with the experimental results, which shows that the RNGk- ϵ turbulence model has a good simulation of the turbulent flow with secondary flow and can accurately reflect the internal Flow, the engineering research has a very good guiding role.

References

- [1] SUDOK, SUNIDAM, HIBARAH. Experimental investigation on turbulent flow in acircular sectioned 90 -degreebend [J] . Experiments in Fluids, 1998(25): 42-49.
- [2] TAYLORAMKP, WHITELAW JH, YIANNESKIS M. Curved ducts with strong secondary motion: velocity measurements of developing laminar andturbulent flow [J]. Journal of Fluids Engineering, 1982(104): 350-359