CFD numerical simulation analysis of small and medium caliber 90 ° circular bend

Wan Kai
School of Electrical Engineering & Automation
Tianjin Polytechnic University
Tianjin, China
wankai606815@163.com

Wang Ping
School of Electrical Engineering & Automation
Tianjin Polytechnic University
Tianjin, China
wangping@tjpu.edu.cn

***Abstract*—Using standard k-ε model with FLUENT software on large diameter CFD numerical simulation of air flow in a 90 ° bent tube, are three dimensional stress field and velocity field in the pipe. To explore non-change law of fully developed pipe flow through CFD numerical simulation on large-diameter flue gas, lays the foundation for analysis and numerical simulation of non-circular cross section.**

Keywords-FLUENT software; Standard k-ε model; Large-diameter 90 ° elbow; CFD numerical simulation.

I. INTRODUCTION

Scholars at home and abroad has researched part of the insufficient development of medium and small diameter tube flow-field distribution and experimental study on the flow measurement methods. With the development of personal computers, academics at home and abroad using CFD part for medium and small diameter non-fully developed pipe flow numerical simulation of flow and flow field distribution measurement methods were studied. China's distribution in the large-caliber short duct flow field numerical simulation have obtained some research achievements, but emphasis on wind tunnel measurements. Numerical simulation of flue gas environment, modeling, rarely turn into the practical application of research. Fluid flow in a curved pipe has long been concerned about the issue that is engaged in the study on the internal flow of workers. With the development of fluid mechanics theory, computer technology, numerical simulation methods, computational fluid dynamics has become an important method for simulation of turbulent flow. This paper adopts standard k-ε model to 90 ° bend three dimensional numerical simulation, which offers guidance function for the non circular cross section numerical simulation.

II. Mathematical model

A. Standard k-ε turbulence model

The standard k-ε model belongs to the eddy viscosity model, which adopts closed RANS equations to solve the model. The used fluid medium is the ambient air, the density ρ to 1.225 kg/m3, kinematic viscosity μ for 1.7894e-5 kg/m.s. Assuming that the air flow rate of 15 m/s the continuous and stable manner flows through the elbow. As the flow rate is small, it can be considered incompressible fluid. By homogenization of the continuity equation and instantaneous Navier-Stokes equations, the Cartesian coordinate system under adiabatic, steady, incompressible fluid flow is governed by the control equation.

B. Continuity equation

$$\partial \bar{u}_i / \partial \bar{x}_i = 0 . \quad (1)$$

C. Equations of motion

$$\frac{\partial}{\partial t}(\rho \bar{u}_i) + \frac{\partial}{\partial x_i}(\rho \bar{u}_i \bar{u}_i) = -\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_i}\left(\frac{\partial \bar{u}_i}{\partial x_i} - \rho \bar{u}_i \bar{u}_i\right) + S_i \quad (2)$$

In the formula, $\bar{u}_i$ is the average velocity of the air, and p is average pressure, and S_i is source term. $-\rho \bar{u}_i \bar{u}_i$ is reynolds stress, By Boussinesgue sticky eddy viscosity model to calculate.

D. Transport equation of the turbulent kinetic energy k

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}\left[\rho u_i k - (\mu + \frac{u_i}{\delta_k})\left(\frac{\partial k}{\partial x_i}\right)\right] = G - \rho\varepsilon . \quad (3)$$

E. Transport equation of Turbulent dissipation rate ε

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}\left[\rho u_i \varepsilon - (\mu + \frac{u_i}{\delta_\varepsilon})\left(\frac{\partial \varepsilon}{\partial x_i}\right)\right] = C_{\varepsilon 1} G \frac{\varepsilon}{k} / - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad (4)$$

In the above equation, $G = \mu_t \left(\frac{\partial u_i}{\partial x_j p} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$, $\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$, The constant in the equation above reference table: $C_\mu = 0.09$, $\delta_k = 1.0$, $\delta_\varepsilon = 1.3$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$.

III. Calculation method

A. Geometric structure and grid generation

First of all, using GAMBIT software based on the physical model for 90 ° bend pipe, the geometric size is shown in Figure 1. For convenience of analysis, the pipeline

is divided into upper straight pipe section, bending section and straight section downstream of three parts. The tube diameter D of 0.4 m, vertical inlet duct length of 0.8 m, a horizontal outlet pipe length of 0.8 m; Origin O is located in bend curvature of the rotary center; And define the mainstream of the inlet section of the curved section as θ = 0 °, and the outlet section of the curved section as θ = 90 °. Meshing software generates the geometry of the computational domain and meshing shown in Figure 2, the grid format Hex / Wedge ,Cooper, hexahedral structured grid unit.

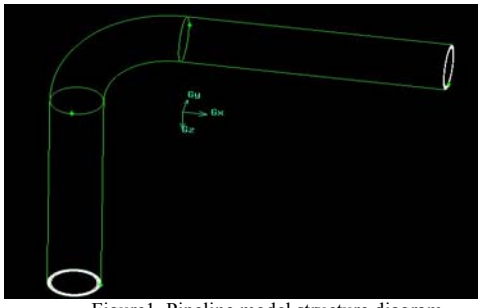

Figure1. Pipeline model structure diagram

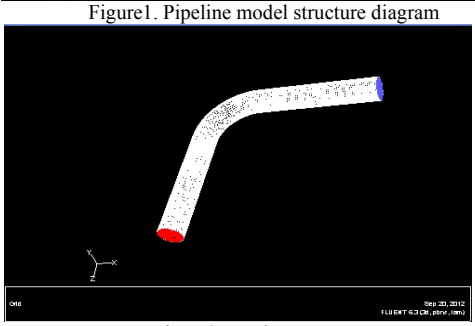

Figure2. Mesh

B. *Boundary conditions*

Boundary conditions at the entrance set by velocity inlet, Inlet velocity for uniform distribution, direction perpendicular to the entrance of bend cross section that is x-z plane; Export based on free-flow conditions. Wall using no slip conditions, body boundary using FLUID boundary.

C. *Calculation methods and parameter settings*

Solver uses pressure-based implicit algorithm, numerical solution of the SIMPLE algorithm for pressure - velocity coupling problem, using second-order accuracy implicit difference scheme, convection item using QUICK difference format, diffusion using central difference scheme.

IV. Calculation result and analysis

A. *Cross section of the residual diagram analysis*

As can be seen iterations 63 times to achieve convergence from Figure 3, which explains that hexahedron mesh can converge easily. And the calculation requires a relatively short time, to be able to reduce the numerical diffusion. For a relatively simple structure, using hexahedral element mesh more easily.

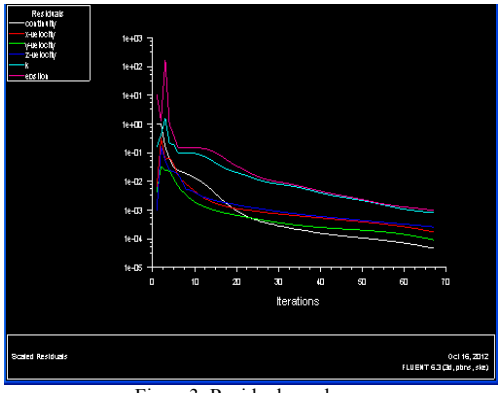

Figure3. Residual graphs

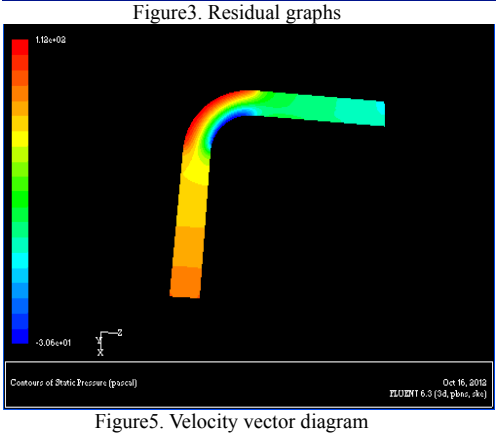

Figure5. Velocity vector diagram

B. *Pressure distribution curve*

The outflow from the exit of the flow of air from the inlet after elbow. Figure 4 shows the inlet velocity of 15m/s, in the elbow, the pressure gradient in the radial direction is large, showing a small near the region of the inner wall surface of the pressure value, the pressure value of the vicinity of the outer wall surface area larger distribution. The main reason for this phenomenon is generated by the bend curvature of the fluid in the flow process due to the centrifugal effect to the radius of curvature larger near the outer wall of the mobile, which causes the plurality of fluid due to push the outer wall surface. Fluid pressure distribution after the outflow pipe to achieve uniform again. In addition, the bend downstream straight pipe pressure value is less than the pressure at the upstream straight pipe, which is mainly caused due to viscous fluid along the pipe wall along the secondary flow and losses arising out of the loss.

Figure4. Pressure diagram

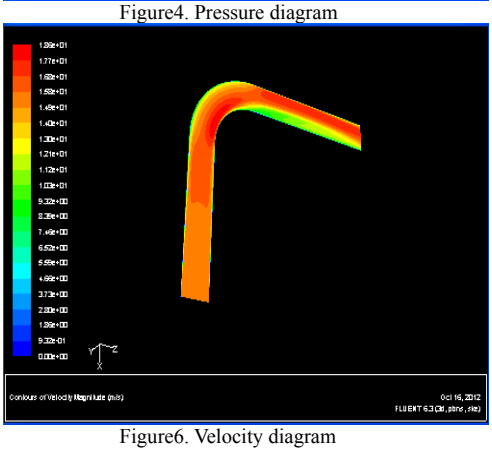

Figure6. Velocity diagram

C. *Velocity vector diagram analysis*

The pressure distribution can be seen in Figure 4, the greater pressure in the vicinity of the outer wall surface of the curved section of the elbow, which the inner wall surface pressure is small; On the contrary, the velocity vector can be seen from Figure 5, in the same position, inner wall nearby speed is bigger, and the speed of the outer wall surface is smaller. Wall velocity in the bend entrance began to increase, high speed flow of fluid to the outside wall, straight pipe downstream of the curved section, and the high-speed fluid appear in the outer wall surface of a straight pipe, and the high-speed fluid region gradually decreases. From Figure 5, As we can all see, bending low velocity area appears near the exit, a large Vortex, which are consistent with experimental results.

V. Conclusion

1) Straight tube segment in the bend before and after, the pressure is evenly distributed; large pressure gradient of the air in the radial direction in the bend segment, which shows a small near the region of the inner wall surface of the pressure value, the pressure value of the vicinity of the outer wall surface area shows larger distribution.

2) Elbow exists a loss of energy, which causing a pressure value of the bending downstream linear segment is smaller than the pressure value of the upstream straight segment.

3) FLUENT standard k-ε turbulence model to calculate the numerical results are in good agreement with the experimental results, indicating that turbulent flow with the secondary flow, the standard k-ε turbulence model has better simulation, which can accurately reflect the elbow internal flow pattern, the numerical simulation of non-circular cross-section pipe.

ACKNOWLEDGMENT

This work was financially supported by Graduate Student Science and Technology Innovation Activity Plan by Tianjin Polytechnic University (12120).

REFERENCES

[1] Yao Zhaohui, Zhou Qiang, Introduction to Computational Fluid Dynamics/(u) Anderson, 1sted, Beijing:Tsinghua University Press,2010.

[2] Jiang Shan, Zhang Jingwei and Wu Chongjian"Based on FLUENT 90 ° Circular Pipe Bending Internal Flow Field Analysis", China ship research, vol 1, pp. 37-41,Mar. 2008.

[3] Zhang Deliang, Computational Fluid Mechanics Course, 1st ed, Beijing: higher education press, October 2010, pp. 469-489

[4] Zhou Junbo, Yang Liu, FLUENT6.3 Analysis of Flow from Entry to the Master, 1st ed, Beijing: China machine press, January 2012, pp. 34-98

[5] Gui Keting, Wang Jun and Wang Qiuying, Engineering Fluid Mechanics, 9th ed, Beijing: Science Press, March ,2011, pp. 86-125