World Scientific Research Journal (WSRJ) ISSN: 2472-3703
www.wsr-j.org

Numerical Simulation of Leakage Flow Field of Natural Gas Elbow Based on FLUENT

Jian Liu[1, a], Xiyang Liu[1, b] and Sheng Chang[1, c]

College of Mechanical and Electronic Engineering, Shandong University of Science and Technology, Qingdao 266590, Shandong China.

[a]2415201321@qq.com, [b]sdust_lxy@163.com, [c]1447965727@qq.com

Abstract: FLUENT software was used to simulate the natural gas elbow. By analyzing the internal flow field of the elbow at different speeds, the change law and main influencing factors of the pressure field inside the elbow were found out.
Keywords: Natural gas elbow, FLUENT, RNG $k-\varepsilon$ model *Numerical Simulation.*

1. INTRODUCTION

With the continuous advancement of sustainable development policies, China has now basically bid farewell to the coal-fired era and entered the oil and gas era. The increasing proportion of natural gas in the energy structure is an important indicator of the improvement of people's living standards and social development. Natural gas is a clean fuel with a calorific value of 34 MJ per cubic meter. It is easy to burn completely and has minimal environmental pollution.

Pipeline transportation is one of the main ways of natural gas transmission. Natural gas pipeline transportation has been rapidly developed in recent years due to its advantages of quick completion and low transportation costs. The elbow is an indispensable part of pipeline transportation. Due to the complex flow field inside the elbow, the FLUENT software is used to simulate the flow of natural gas inside the elbow when it leaks. The internal flow field is analyzed and the change law is analyzed.

2. CALCULATION MODEL

2.1 Physical Model Establishment and Meshing

The leakage model of natural gas elbow is shown in Fig 1. It includes the internal part of the elbow and the two parts of the leakage spray zone. In the model, the pipe diameter is 0.2m, the length of two straight pipes is 0.2m and the radius of the pipe is 0.15m.The leak is directly above the bend and the length of the leak stream area is approximately 0.11m.The physical model of this article is established through solid works and imported into the workbench to divide the grid The grid generation graph is shown in Fig 2, with a total of 45,400 grids.

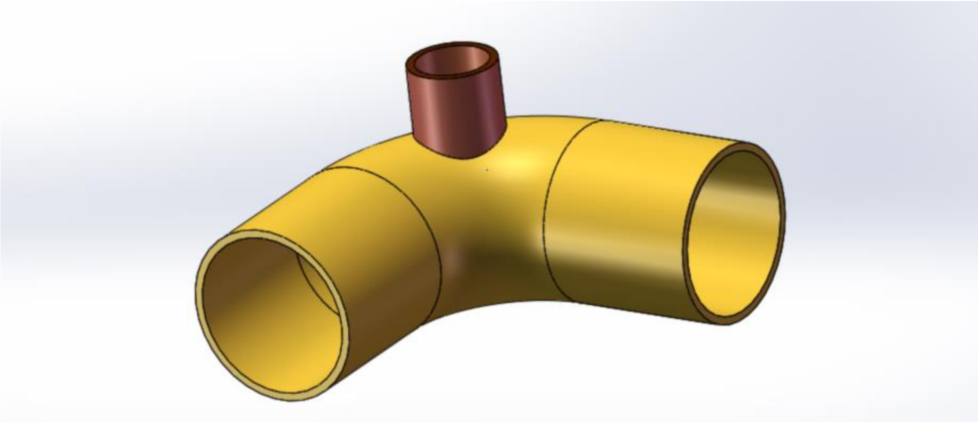


Fig 1. Natural gas elbow model

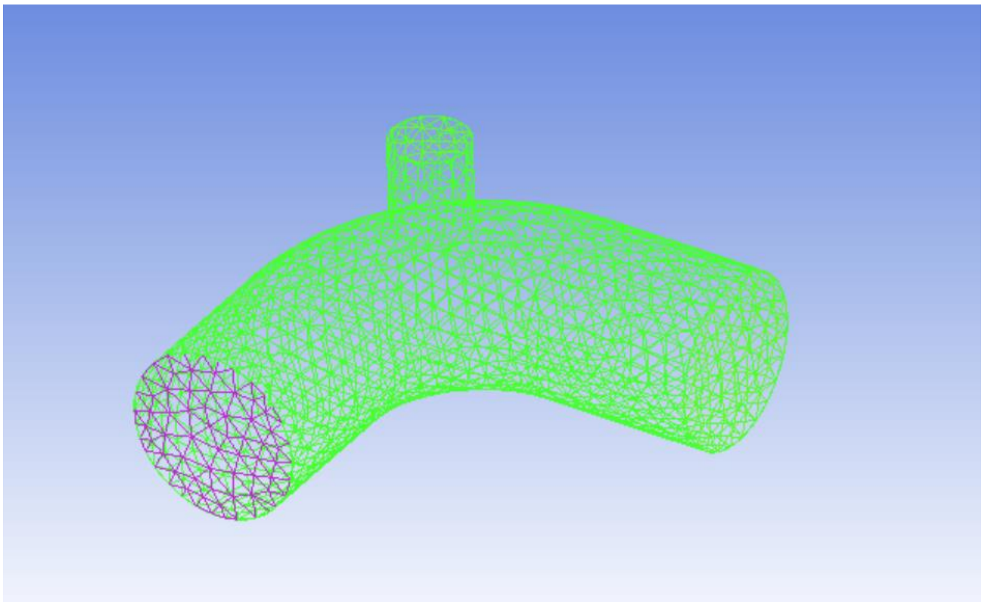


Fig 2. Elbow grid model

2.2 Calculation Model Equation

This article studies the nature of the flow field of natural gas in a bend. It can be assumed that the fluid flow is an incompressible flow. The Navies-Stokes equations and the continuity equations are homogenized to obtain the control equations in the rectangular coordinate system.

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \overline{u_i}) + \frac{\partial}{\partial x_j}(\rho \overline{u_i}\,\overline{u_j}) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu \frac{\partial \overline{u_i}}{\partial x_j} - \rho \overline{u_i'}\,\overline{u_j'}\right] + S_j \tag{2}$$

In the formula ρ -Fluid density, kg/m^3

$\overline{u_i}$ -The average velocity of liquid in the x direction, m/s

$\overline{u_j}$ -The average velocity of the liquid in the y direction, m/s

p -pressure, MP

$\rho\overline{u_i'}\,\overline{u_j'}$ -Reynolds stress, Pa

S_j -Source project

RNG $k-\varepsilon$ model considers the rotation and swirling flow conditions in the average flow, so it is more reliable when analyzing the flow conditions. This model also adapts to the conditions of high and low Reynolds number and combines with the wall function processing method to more reasonably simulate the flow field data of natural gas elbow.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\alpha_k\mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon\mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon+S_\varepsilon \tag{4}$$

In the formula $\alpha_k - k$ Turbulence Brandt Number of Equations

$\alpha_\varepsilon - \varepsilon$ Turbulence Brandt Number of Equations

$G_k -$ Laminar flow gradient resulting turbulent kinetic energy, NM

$G_b -$ Buoyant turbulent kinetic energy, NM

K-Turbulence kinetic energy

2.3 Boundary Condition Setting and Algorithm Selection

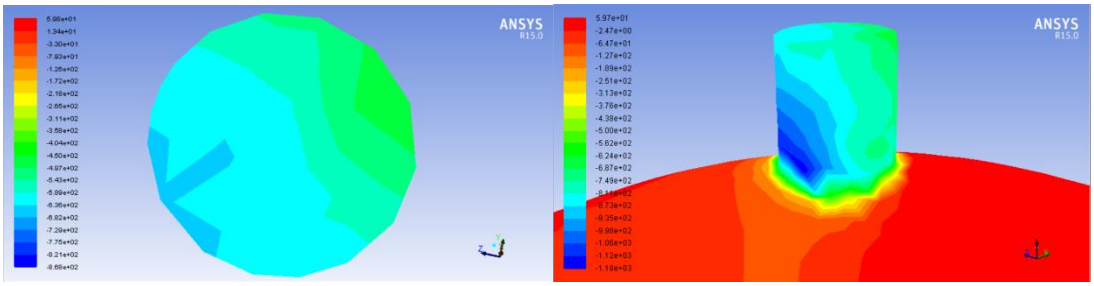
The fluid object studied in this paper is the natural gas in the bend. The main component of natural gas is methane. The density was 0.72 Kg/m^3 at 0℃ and 101 kappa, and the relative density was 0.5548.Using velocity inlets to analyze gas flow rates perpendicular to the cross section at the inlet of natural gas elbows at 10, 15 and 20 m/s, respectively. When the temperature is set to 25℃, the outlet pressure is 0 for simulation. Finally, the SIMPLE algorithm is used for calculation.

3. FLUENT FLOW FIELD SOLUTION

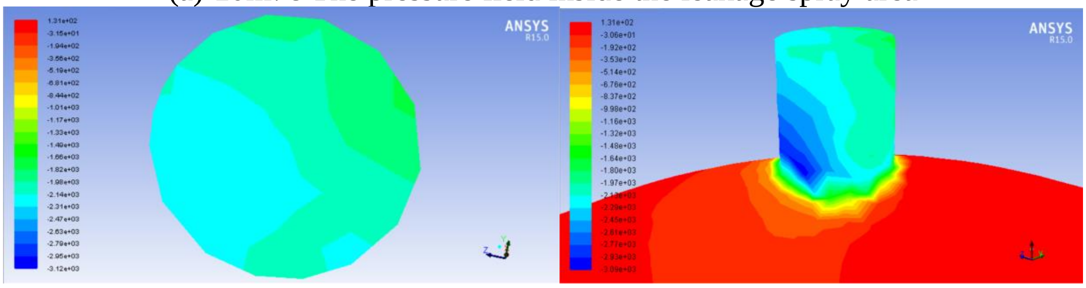
When a leak occurs at the elbow, there is a large difference in pressure between the inside and outside of the pipe and the natural gas with a faster flow rate in the leakage spout area is ejected. Outside the leakage area of the pipe, the pressure difference is relatively stable, so this article focuses on the change of pressure and velocity of the flow field at the leak hole.

3.1 Pressure Field Analysis

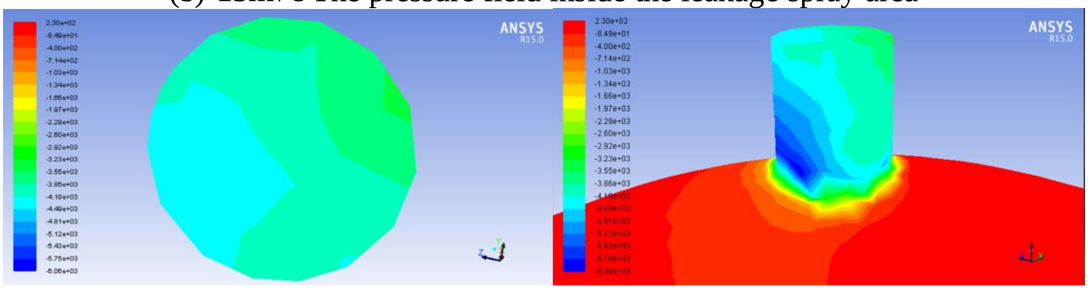
The flow rate at the inlet of the natural gas elbow is taken as 10, 15 and 20 m/s, respectively, the pressure field distribution is shown in the figure below.



(a) 10m/s The pressure field inside the leakage spray area



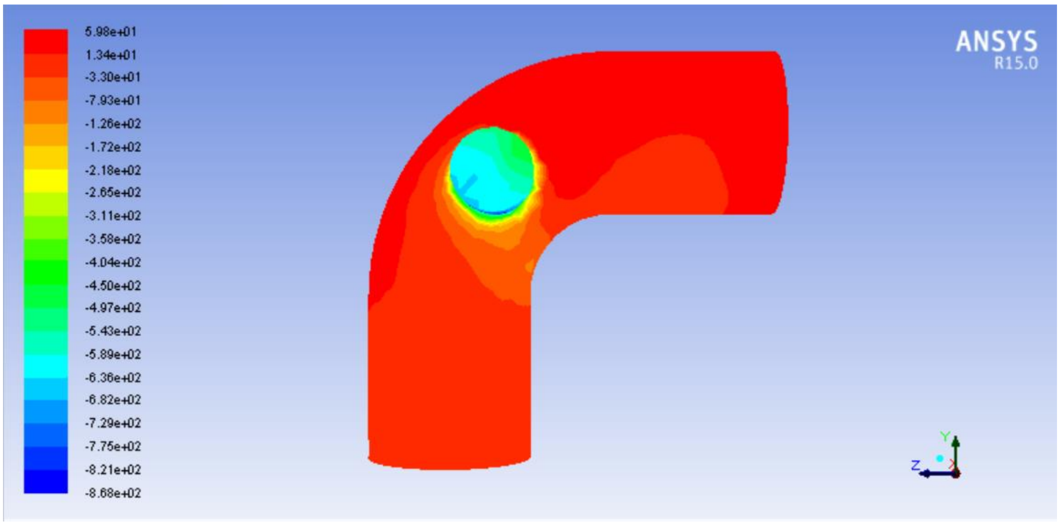
(b) 15m/s The pressure field inside the leakage spray area



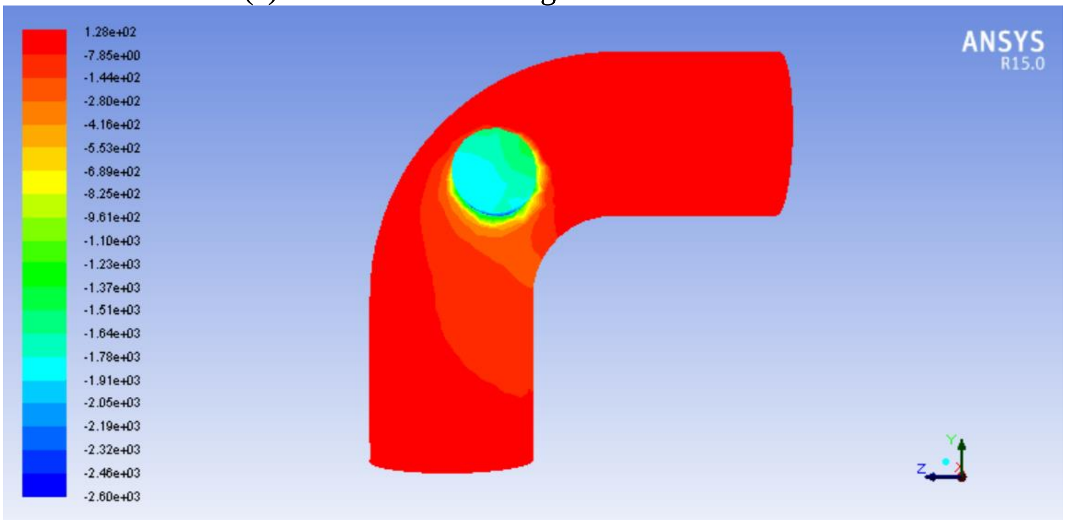
(c) 20m/s The pressure field inside the leakage spray area

Fig 3. Pressure field diagram inside the leakage spray area

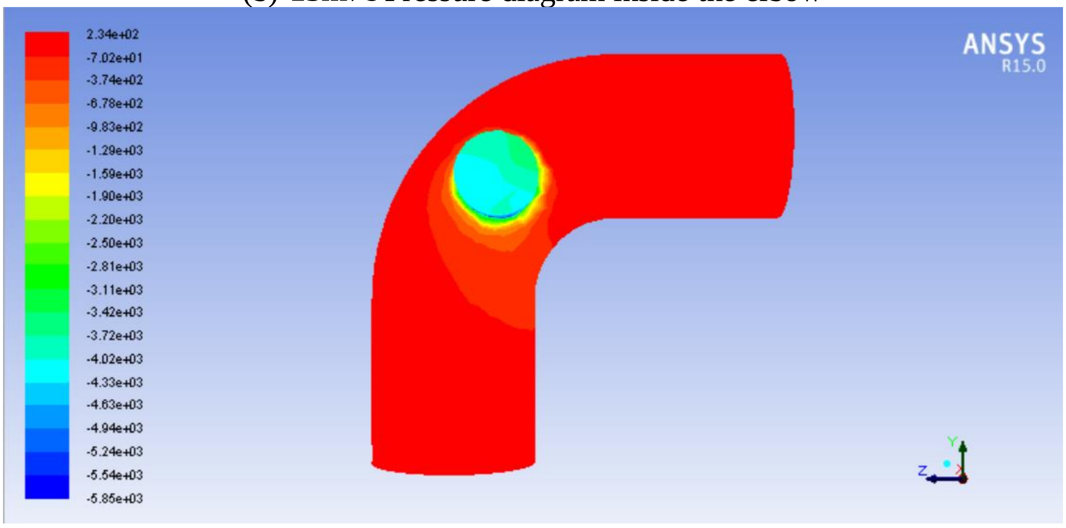
As can be seen from Figure 3, when the natural gas flow rate increases from the initial 10 m/s to 15 m/s, the pressure at the outlet of the elbow increases significantly; when the flow rate increases from 15 m/s to 20 m/s, the pressure at the outlet increases more slowly. This shows that when the speed is relatively large, a small increase in speed will not cause a significant increase in the internal pressure of the elbow.



(a) $10m/s$ Pressure diagram inside the elbow



(b) $15m/s$ Pressure diagram inside the elbow



(c) $20m/s$ Pressure diagram inside the elbow

Fig 4. The pressure field inside the elbow

It can be seen from Fig 4 that the pressure inside the natural gas elbow decreases in gradient because of the energy loss during the transportation of natural gas.

4. CONCLUSION

(1) In this paper, the RNG $k-\varepsilon$ model in FLUENT software is used to numerically simulate the fluid inside the natural gas elbow, which can better simulate the secondary flow inside the elbow. In future numerical simulations, other turbulence models can be attempted to make the secondary flow simulation more accurate.

(2) When the initial velocity of the fluid in the natural gas elbow is different, the following conclusions are obtained when analyzing the pressure field: When the speed is relatively large, the effect of increasing the speed on the pressure inside the elbow is not obvious; the velocity on the inner wall surface of the elbow is larger, the pressure on the outer wall is larger.

REFERENCES

[1] Prashant K S, Mullica A N. Computational investigation of performance characteristics in a C-shape diffusing duct [J].International Journal of Advances in Engineering & Technology, 2012 3 (1): 129-136.

[2] Faun Wang. Computational Fluid Dynamics Analysis - CFD Software Principle and Application [M] .Beijing: Tsinghua University Press, 2004: 7-13, 121-122.

[3] Shan Jiang, Jingle Zhang, Chongjin Wu .Analysis of Internal Flow Field of 90 Degree Circular Bend Pipe Based on FLUFNT [J].Chinese Ship Research, 20083(1):37-41.

[4] Faun Wang. Computational Fluid Dynamics Analysis - CFD Software Principle and Application [M] .Beijing: Tsinghua University Press, 2004: 7-13, 121-122.

[5] Mohammed A. Al-Yaris, Basel F. Abu-shark. CFD prediction of oil-water phase separation in 180° bend. [J]Asian Transactions on Engineering, 2011, 1(5):63-67.