Study on numerical simulation of gas pipeline speed regulating detector

Li Xiaolong¹, Meng Tao¹, Ma Yilai¹, Tan Zheng², Chen Jinzhong¹

¹China Special Equipment Inspection and Research Institute, Beijing 100029, China
²School of mechanical engineering, Changzhou university, Changzhou, 213164, China
xiaolongtlee@163.com

Abstract. In order to control the running speed of the detector in the gas pipeline, the flow field calculation method of the detector in the gas pipeline under the discharge state is studied. Based on the operating principle of the detector in the gas pipeline, the model of the discharge hole of the detector is constructed by using the standard orifice flowmeter. Based on computational fluid dynamics theory, the application of computational fluid dynamics software ICEM CFD and provided ANSYS CFX to generate flow field model and a numerical simulation was carried out, with fluid flowing through the discharge of the flow field simulation, the discharge state of gas pipeline detector in flow field, achieved under different aperture of gas flowing through the standard orifice plate before and after the pressure field, velocity field of numerical analysis. The results show that using the standard hole as the detector model in the gas pipeline under the discharge state has certain guidance, and it is of certain significance to further study the velocity control problem of the detector in the gas pipeline.

1. Introduction
As the key technology of the inner detector, the velocity control problem of the inner detector in the long-distance pipeline has been studied by many experts, scholars and scientific research institutions. Lima established the transient model of the internal detector under the two-phase flow model in 1999. Although it was only for the pig, the establishment of the transient model was still of reference value in the establishment of the transient model of the controllable fast pig. South Korea’s Nguyen and others since 2001, carrying out a research into the internal detector speed control [1-3], they will be long distance natural gas pipeline within the medium as a one dimensional isothermal quasi steady state flow, integrated detector driven hydrodynamic model, the detector within the discharge power model and motion detector itself detector in dynamic model is deduced. Hosseinalipour S M, etc was studied in the gas pipeline detector in the transient simulation method [4], the model assumes that the gas fluid of the pipe as the ideal, the slip and cross grid for discretization model to solve and test verification, the results show that the method of internal detector motion prediction is consistent with the experimental results, show that the method is established in the transient model of detector is effective. In 2006, Florian from ROSEN company in Germany published an article introducing a simplified form of transient model of controllable velocity internal detector running in natural gas pipeline. This model is a differential equation related to the running speed, operating pressure and bypass region pressure difference of the internal detector, and verified the effectiveness of this simplified model [5].

In this paper, fluid dynamics software is used to simulate the running speed of the detector in the gas pipeline. In shear stress transport turbulence model simulation environment, by setting the
corresponding boundary condition in the simulation of the detector in a certain gas velocity of flow field and pressure changes, this paper analyzes the different discharge hole size and fluid velocity within the detector under the differential pressure flow field before and after the internal driving force of the detector inside gas pipeline with related pipeline operation factors, is advantageous to the analysis of capacity control detector and make it to the appropriate speed.

2. Modeling

2.1 The running rule of internal detector in natural gas pipeline
When the inner detector moves in the gas transmission pipeline, the flow field of the gas in the pipeline will provide its forward power. Relying on the pressure transmitted in the pipeline as the driving force, the inner detector will move in the gas transmission pipeline. Moreover, the gas flow field in the pipeline will also change due to the presence of the internal detector. In order to control the running speed of the detector, a discharge device is designed inside the detector, which can be used as the action signal of the discharge valve switch by detecting the running speed of the detector. When the detector runs too fast, the outlet hole area should be increased in time to increase the gas flow through the outlet hole and reduce the pressure difference between the upstream and downstream of the detector, so as to reduce the operating speed of the detector. Similarly, when the velocity of the detector does not reach a moderate value, the valve will be closed by reducing the amount of gas passing through, and the area of the discharge hole will be reduced. Finally, the velocity of the detector will increase and reach the expected value.

2.2 The model of internal detector discharge state in natural gas pipeline
The actual structure of the in-pipeline detector is very complex, and the in-pipeline detector designed by different countries is also different. Therefore, it is very difficult to build the calculation model completely in accordance with its real model, and it is unnecessary to increase the difficulty for the simulation task. The computational model can be simplified according to the feature structure of the real model. It should focus on the simple model, retain the structural components that have significant influence on the flow field, and ignore the secondary structural features. In this way, the authenticity of the surrounding flow field can be guaranteed, and the computation amount can be reduced. Combined with computational fluid dynamics, grid technology and rigid body dynamics, the gas flow field around the detector in the gas pipeline is simulated when the detector is in steady state. In this paper, based on the above characteristics and simulation requirements, the relationship between the opening area and the pressure difference was calculated under the same boundary conditions. The standard orifice flowmeter was selected as the detector model to simulate the bypass area and analyze. A circular pipe of equal diameter was simulated as a gas pipeline, and the origin of rectangular coordinate system was set at the entrance dot, and the z-axis was along the flow direction of gas. The fluid is incompressible, ignoring gravity. The total length is 6.8m, the pipe diameter is 660mm, and the detector length is 0.8m. At the aperture, six kinds of models with the open area of 30%, 25%, 20%, 15%, 10% and 5% were established respectively. The inlet to the aperture is 1m, and the aperture to the outlet is 5m. The key of numerical simulation is first accurate mathematical model and second effective numerical solution method. The model is put into ICEM to divide unstructured grid, which has better adaptability. A total of 473,449 grids were divided. The quality of the grids ranged from 0 to 1, and the closer to 1, the better. Among them, there were 374,584 grids with a mass of 0.9~1, accounting for 79.1%, 48,879 grids with a mass of 0.8~0.9, accounting for 10.3%, and 0.4~0.8, accounting for 10.6%. It can be seen that the quality of grid division is good and meets the requirements of flow field calculation. The model is generated by ICEM software, as shown in fig.1.
3. The flow field analysis

Fluid flow is governed by the law of conservation of physics. The basic conservation laws include the law of conservation of mass, the law of conservation of momentum and the law of conservation of energy. If the flow is turbulent, the system is subject to the additional turbulent transport equation. Any flow problem must satisfy the law of conservation of mass. The law of momentum conservation is also a fundamental law that any flow system must satisfy. This law can be expressed as follows: the rate of change of the momentum of the fluid in the infinitesimal body is equal to the sum of the various forces acting on the infinitesimal body by the outside world. The basic equation for the analysis of compressible turbulent flows is the Reynolds average navier-stokes equation.[6-7]

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]  
(1)

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j + \delta_{ij} P) = \frac{\partial}{\partial x_j} \left[ \tau_{ij} \right] \quad \text{for} \quad i,j = 1, 2
\]  
(2)

\[
\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} (\rho u_i H) = -\frac{\partial}{\partial x_j} \left[ \tau_{ij} u_j + (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} - q_i \right]
\]  
(3)

Where, \( t \)-time, \( u_i \)-position vector, \( P \)-Fluid pressure, \( \rho \)-Fluid density, \( \kappa \)-Turbulent kinetic energy, \( \mu \)-Molecular viscosity, \( E \)-Stagnation internal energy, \( H \)-Stagnation enthalpy, \( H = h + \frac{u_i u_j}{2} \), \( e \)-Internal energy, \( h \)-enthalpy, \( \tau_{ij} \)-The sum of the molecular Reynolds stress tensors, \( q_i \)-The sum of molecules and turbulent heat flow vectors. 

For an ideal gas, the relationship is described by the ideal gas law [8].

\[
\rho = \frac{\alpha \rho + p_{ref}}{R_0 T}
\]  
(4)

Where, \( \omega \)-Gas molecular mass, \( p_{ref} \)-Reference pressure, \( T \)-Temperature, \( R_0 \) usually, 8.314J/(mol*k).

Viscous flow can be divided into laminar flow and turbulent flow. Laminar flow refers to the smooth linear motion of the particle along the direction parallel to the tube axis when the fluid is flowing in the tube. The SST turbulence model based on k-ohm model was used to integrate the advantages of k-ohm model in the near-wall area and the advantages of k-ohm model in the far field. The model was obtained by multiplying the k-ohm model and the standard k-ohm model by a mixed function and then adding them. In this turbulent state, the system follows equation (5):

\[
v_j = \frac{\alpha_k}{\max(\alpha_i, \text{SF})}
\]  
(5)

Where, \( v_j \)-Eddy viscosity, \( F \)-Mixed function, \( S \)-Strain rate constant, \( \alpha_i \) Usually 5/9.

The most robust setting is adopted for the boundary conditions, that is, the inlet boundary conditions are set for the normal velocity, the outlet boundary conditions are static pressure and the appropriate wall boundary conditions.

4. Numerical calculation

In order to facilitate the calculation, the model is simplified. The following hypothesis is proposed:
4.1 The determination of domain parameters and boundary condition parameters

Medium parameters and physical parameters are selected according to actual parameters. In CFX-Pre, domain parameters are set as shown in table 1.

| Parameter       | Setting       |
|-----------------|---------------|
| Fluid type      | Ideal gas     |
| Flow of the domain | Steady state |
| Reference pressure | 0.1Mpa       |
| Turbulence      | Shear stress transmission |
| Heat transfer   | Isothermal    |

Boundary condition setting. All CFD problems can be solved correctly only when reasonable boundary conditions are given. In these four models, uniform boundary conditions (table 2-3). Flow inlet boundary conditions: under the subsonic condition, the normal velocity is set as 5m/s respectively.

| Parameter       | Setting       |
|-----------------|---------------|
| Boundary types  | Outlet        |
| Flow pattern    | Subsonic      |
| Mass and motion | Static pressure |
| Static pressure | 5Mpa          |

4.2 Simulation result

The medium is assumed to be air and the temperature is 25℃. By analyzing the flow of fluid before and after the orifice plate, the distribution of pipe velocity field and pressure field of the six models at 5m/s inlet velocity was obtained, and YZ plane was taken, as shown in figure 2. As can be seen from figure 2 velocity flow field distribution diagram, axial velocity dominates the mainstream, so axial velocity is taken as the main basis in design. Before the fluid flows into the discharge hole, the velocity changes little, indicating that the fluid flow field is relatively stable in the upstream. After the outflow of the spillway hole, the change of velocity was obviously accelerated. From the velocity distribution of the whole flow field, the downstream change of the inner detector is much more drastic than that of the upstream, and the force on the detector is relatively complex. Therefore, vulnerable components such as sensors should not be installed in the downstream flow field.
Figure 3 shows the relation between the leakage opening and the pressure and pressure difference before and after the detector under the boundary condition of inlet flow rate of 5m/s and outlet pressure of 5Mpa. According to Figure 3(a), if the downstream outlet pressure is to be maintained as the discharge opening increases, the upstream pressure needs to increase. Similarly, according to Figure 3(b), if the downstream outlet pressure is to be maintained as the discharge opening increases, the downstream pressure of the detector needs to decrease. According to the analysis in Figure 3(c), when the opening is less than 10%, the pressure difference decreases sharply with the increase of the discharge opening, and is greatly affected by the discharge opening, and the change is obvious. When the opening is greater than 10%, the pressure difference decreases with the discharge opening but tends to be gentle. The simulation results show that by adjusting the discharge area, the pressure difference before and after the detector can be effectively adjusted, and then the operating speed of the detector in the gas pipeline can be controlled.
Fig. 3 The relationship between discharge opening and pressure, pressure difference under the condition of inlet velocity of 5m/s and outlet pressure of 5MPa

5. Conclusion
Three-dimensional CFD is applied in this article provided theory and computational fluid dynamics software CFX mean field on gas pipeline detector in discharge status of research, establishes the downstream field of the detector discharge physical model and turbulent model, and a numerical simulation was carried out, the velocity field, pressure field distribution is obtained and the relationship between the discharge hole size and the differential pressure, as the actual experiment in the detector running speed and the pipeline detector aided design provides technical support.

Acknowledgments
The work was financially supported by China Special Equipment Inspection and research institute youth fund program “Study on key technology of velocity control of detector in natural gas pipeline” (2018-youth16)

References
[1] Nguyen T T, Yoo H R, Rho Y W, et al. Speed control of PIG using bypass flow in natural gas pipeline[C]. 2001 IEEE ISIE. Pusan Korea, 2001: 863-868.
[2] Nguyen T T, Kim D K, Rho Y W, et al. Dynamic modeling and its analysis for PIG flow through curved section in natural gas pipelines [C]. Proceedings of 2001 IEEE International Symposium on Computational Intelligence Robotics and Automation, 2001: 492-497.
[3] Nguyen T T, Kim S B, Yoo H R, et al. Modeling and simulation for pig with bypass flow control in natural gas pipeline [J]. KSME International Journal, 2001, 15(9): 1302-1310.
[4] Hosseinalipour S M, Khalili A Z, Salimi A. Numerical simulation of pig motion through gas pipeline[C]. 16th Australasian Fluid Mechanics Conference, Australia, 2007: 971-975.
[5] Rahe D F. Optimizing the Active Speed Control Unit for In-Line Inspection Tools in Gas [C]. 2006 International Pipeline Conference. American Society of Mechanical Engineers, 2006:377-383.
[6] Li D N, YUAN Jiang-nan, YANG Yi-min. Velocity calculation and analysis of in-pipe robot power self-supported by fluid[J].Chinese Journal of Mechanical Engineering, 2007, 8(43):17-21.
[7] YUAN Xin. Selection of two-equation turbulence model for compressible viscous flow [J]. Journal of Engineering Thermophysics, 1998,19(4):427-430.
[8] ZHOU Zhi-jun, LIN Zhen, ZHOU Jun-hu. Application of different turbulent models in calculation of flow resistance in pipelines and comparison thereof [J]. Thermal Power Generation, 2007(1):18-22.