Research on flow field characteristics of flowmeter fairing based on CFD

Y. Xiang, M. Y. Qin, Y. F. Xing, W. H. Zhou, X. Liu*

Chongqing Academy of Metrology and Quality Inspection, Chongqing, China
E-mail (corresponding author): 273158559@qq.com

Abstract

In this paper, Under complex pipeline conditions, the fluid flow field is seriously distorted, which affects the measurement accuracy of the flowmeter. As one of the main components of the flowmeter, the rectifier has an important impact on its measurement accuracy and reliability. Aiming at this problem, the simulation experiments of rectifiers with different structures are carried out in this paper. Firstly, combined with the working principle and structural composition of the turbine flowmeter, the factors affecting the measurement accuracy of the flowmeter are analyzed. Then, the structural model of the rectifier and the pipeline is established, and the flow field distribution in three cases without rectifier, with honeycomb rectifier and with arc rectifier is studied by CFD simulation.The simulation results show that compared with no rectifier, the honeycomb rectifier and the circular arc rectifier have good performance in effectively reducing eddy and turbulent flow, providing more stable and repeated velocity profiles and further improving the flow field distribution. Under the same conditions, the rectification effect of the honeycomb rectifier on the fluid flow field is more significant.This work is very useful for accurate measurement of turbine flowmeter, especially for the complex pipeline conditions.

1. Introduction

Flow monitoring plays an important role in modern industrial production. It is an important measurement parameter in scientific research and production process[1-2]. With the establishment of the project of natural gas transmission from West to East, the gas flowmeter has received extensive attention, and its measurement accuracy, structural reliability and operation convenience have been greatly improved.

The working principle of the turbine flowmeter is that the fluid impacts the mechanical parts of the flowmeter, drives the parts to rotate, and calculates the flow rate of the fluid per unit time by measuring the rotational speed of the parts[3-4]. However, due to the complexity of the flow field, the flow velocity distribution of the fluid is uneven, which seriously restricts the accuracy of the measurement results of the flowmeter. Therefore, rectifying the flow field has become the top priority[5-6].

Chen[7] based on theoretical analysis, a three-dimensional flow field CFD model of the turbine flow sensor.The cavitation tunnel experiment is performed to obtain the sensor characteristics. simulation and experiment results is analyzed and the feasibility of the CFD simulation of the sensor flow field is proved. Mohammad Amin Alaeddin[8] with the aid of UCCF analytical model, the effect of the mentioned parameters including flow Reynolds number, the straight pipe length at the upstream of the flowmeter and the roughness of the pipe, on the performance of the UCCF were investigated. Yao[9] used CFD to numerically simulate the flow field inside the measuring pipe, and designed a rectifier to improve the obvious secondary flow and eddy current caused by the elbow, and reduce the measurement error of the ultrasonic flowmeter. Shao[10] proposed an optimization idea to reduce the separation area and improve the guide effect of the guide vane. The metering performance of the gas turbine flowmeter has been significantly improved. Wang[11] analyzed the characteristics of the internal flow field before and after the structure optimization of the turbine flowmeter. The results showed that the sudden change of pressure drop in the area of the front rectifier and the rear guide fluid, the vortex structure and the backflow phenomenon at the tail of the rear guide fluid are the main factors that affect the measurement performance of the flowmeter. main mechanism. Yuan[12] designed a combined rectifier and used CFD to study the effect of fan blade angle and cavity length on the flow field of the flowmeter. Better performance in terms of stable velocity profiles.

Aiming at the key problem of flow field rectification, this paper analyzes the working principle and structural composition of the turbine flowmeter, takes the rectifier of the turbine flowmeter as the research object, and adopts the method of combining finite element simulation and numerical analysis to carry out the flowmeter after rectification. The flow field characteristics research of , obtained the flow field velocity and pressure distribution under different rectifier structures, and analyzed the reasons for their formation. This work is very useful for the accurate measurement of gas turbine flowmeters, especially for complex pipeline conditions.

2. Flow meter working principle and model establishment

2.1 Flow meter working principle

Turbine flowmeter is a velocity-type flow measuring instrument. Its working principle is that when the fluid flows through the flowmeter, the power of the flowing fluid drives the turbine blades to rotate, and its rotational speed is proportional to the volume flow, and then obtain the flow value of the fluid in the pipeline.

Under ideal working conditions without considering the frictional resistance torque and fluid resistance torque of the impeller, the flow equation of the flowmeter can be expressed as:

$$Q = \frac{A 2\pi r f}{N \tan\beta}. \tag{1}$$

Q is the volume flow, f is the rotation frequency of the impeller, r is the radius, A is the flow cross-sectional area, β is the blade installation angle, and N is the number of blades.

The relationship between the rotation frequency of the blade and the average fluid velocity can be expressed as:

$$f = \frac{vN\tan\beta}{2\pi r}. \tag{2}$$

The impeller angular velocity ω can be expressed as:

$$\omega = \frac{v\tan\beta}{r}. \tag{3}$$

It can be seen from the above formula that the rotational angular velocity of the impeller is proportional to the average velocity of the fluid entering the impeller, which in turn is proportional to the volume flow of the fluid. Therefore, the velocity distribution before the fluid enters the impeller directly affects the measurement performance of the turbine flowmeter, so the fluid needs to be rectified.

2.2 Model establishment

The structure of the flowmeter is mainly composed of the front rectifier, the impeller, the rear guide body and the shell. This paper mainly studies the rectification effect of the rectifier on the fluid in the pipeline. The model is established as shown in figure 1. The simplified model is mainly composed of elbow, rectifier and impeller. composition.

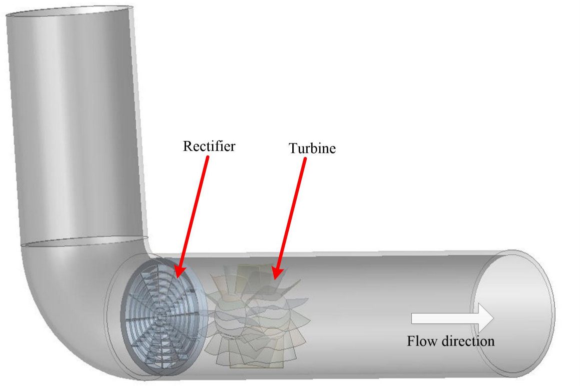

Figure 1: Simulation model.

It is common for a flow blocker to be installed upstream of the flowmeter. The existence of elbows and valves will generate fluid in the pipeline and change the flow field. As the fluid inlet of the flowmeter is closer to the fluid outlet of the blocker, the flow The more complex the flow field distribution inside the measuring pipe, the more complex the flow field distribution, which will affect the measuring accuracy of the flowmeter. In order to simulate the unstable flow field in the actual working condition, investigate the rectification effect of the rectifier, introduce the design of the elbow, and set a 90° elbow in front of the rectifier.

In this paper, two kinds of rectifiers, cellular type and arc type, are used for analysis, and the structure of the rectifier is shown in figure 2.

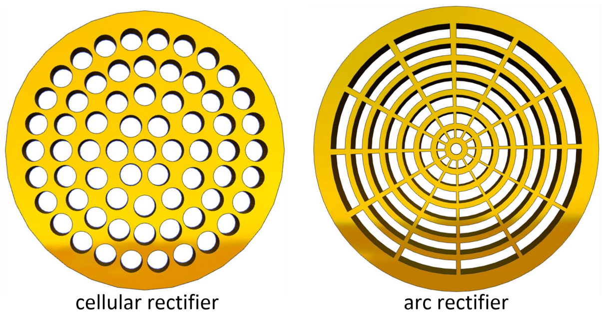

Figure 2: Rectifier model.

3. Simulation analysis

3.1 Flow field simulation of elbow without rectification

In this paper, the distribution of the fluid flow field in the elbow under the condition of no rectification is firstly studied, and a 3D geometric model is established by UG. The diameter of the pipe is 50mm, and the mesh independence is verified. The unstructured mesh of the tetrahedron type is used. The boundary conditions for the Fluent solver are set as follows: the inlet is set to the velocity inlet, the outlet is set to the pressure outlet, set to one atmosphere pressure, and the pipe wall is the wall. Combined with the actual flow conditions and the

Reynolds number and other indicators, it can be seen that the fluid in the pipeline is turbulent, so the turbulent flow model is selected as the fluid calculation model. Since the turbulent kinetic energy k and the turbulent dissipation rate epsilon parameters are not directly input, the turbulence parameters are set to Intensity and Hydraulic Diameter.

After the parameter setting is completed, the solution calculation is carried out to study the pressure change and velocity of the fluid in the pipeline. The pressure cloud map and velocity cloud map in the pipeline are extracted in the post-processing module of CFD, and the results are shown in the figure.

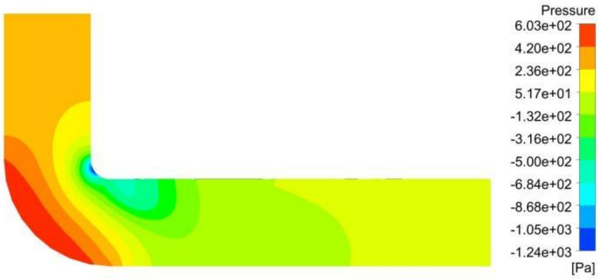

Figure 3: Pipe velocity contour without rectifier.

It can be seen from figure 3 that the pressure change of the fluid in the pipeline can be divided into four parts, namely the pressure-stabilizing part at the front of the elbow, the high-pressure part at the outer wall of the elbow, the negative pressure part at the inner wall of the elbow and the gradient part at the rear end of the pipeline. When the fluid flows through the elbow, due to the flow resistance of the outer wall of the elbow, the pressure at the outer wall of the elbow increases suddenly. With the increase of the length of the distance from the elbow, the influence of the elbow is gradually eliminated and the fluid returns to the fully developed turbulent state.

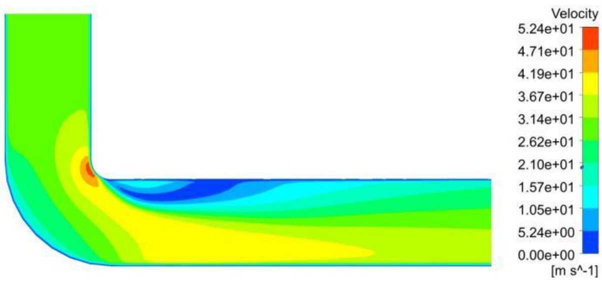

Figure 4: Pipe pressure contour without rectifier.

It can be seen from figure 4 that the flow direction of the fluid changes abruptly after flowing through the elbow. The flow velocity at the wall decreases in layers. As the fluid continues to move forward along the long straight pipeline, the gas and fluid particles interact with each other in the main flow direction of the pipeline, and the flow becomes gradually stable.

3.2 Simulation analysis of rectification effect of cellular rectifier

For the pipeline model with the rectifier, a hybrid meshing method combining structured and unstructured is used, the model is cut into pieces, the model is divided into two parts, the pipeline and the rectifier, and the mesh is divided into two parts, and the key parts are meshed. Encrypted processing. The schematic diagram of the model grid is shown in figure 5.

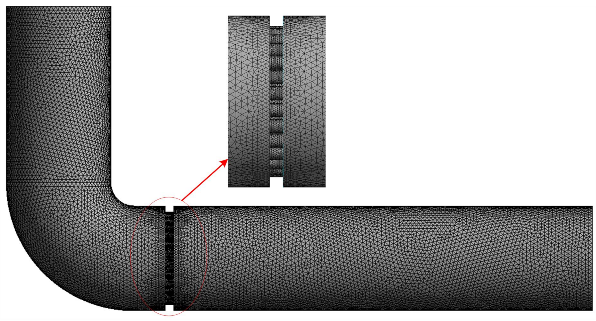

Figure 5: Meshing scheme used for discretization of the flow domains.

The fluid-structure interaction simulation was performed on the pipeline model with the rectifier added, and the pressure and velocity changes in the fluid domain after rectification were analyzed. The results are shown in the figure.

Figure 6: Pipe pressure contour with cellular rectifier.

It can be seen from figure 6 that the rectification effect of the honeycomb rectifier is obvious, which can quickly eliminate the pressure disturbance caused by the elbow. After being rectified by the rectifier, the fluid pressure changes gradually and smoothly, basically eliminating the high-pressure region, and finally returns to a fully developed turbulent state at the downstream 1D.

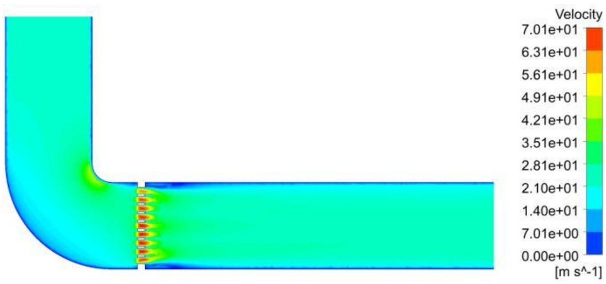

Figure 7: Pipe velocity contour with cellular rectifier.

It can be seen from figure 7 that after the fluid passes through the cellular rectifier, the velocity disturbance caused by the elbow can be quickly eliminated, and the velocity distribution in the central area near the rectifier fluctuates slightly, and there is no low-speed area. At the same time, the velocity distribution near the pipe wall is also The velocity curve is relatively close to the fully developed, and basically presents the fully developed initial state when it reaches the downstream 1D.

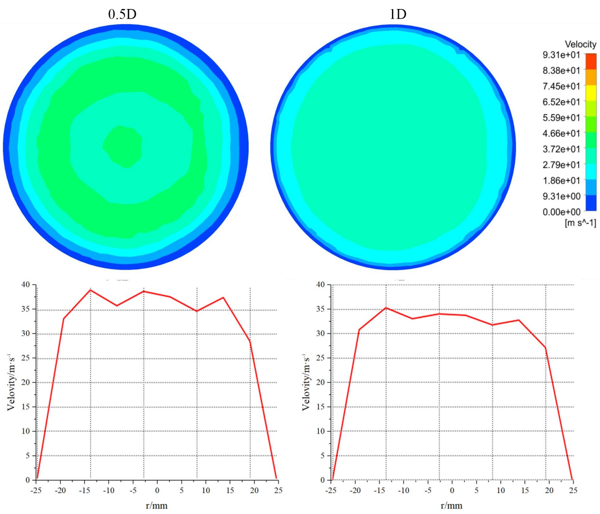

Figure 8: Velocity contour map of pipeline section with cellular rectifier.

It can be seen from the schematic diagram of the flow velocity distribution of each section downstream of the elbow of the cellular rectifier in Figure 8 that after the cellular rectifier is installed in the pipeline, after the fluid passes through the cellular rectifier, the cross-sectional flow field at 0.5D has greatly eliminated the asymmetry of the original flow field A relatively symmetrical trend has been formed, and the turbulent distribution of the flow field has been significantly improved, but the flow field still has relatively obvious velocity fluctuations. Then, with the development of the flow, the velocity distribution becomes uniform, the velocity fluctuation gradually weakens, and finally reaches a stable development state at 1D.

3.3 Simulation analysis of rectification effect of arc rectifier

The rectification effect of the arc rectifier is simulated and analyzed in the same way. The simulation conditions remain unchanged. The pressure and velocity changes of the fluid domain inside the pipeline can be obtained through the simulation. The results are shown in the figure .

Figure 9: Pipe pressure contour with arc rectifier..

It can be seen from figure. 9 that the arc rectifier plays an obvious role in eliminating the flow field disorder. It can quickly eliminate the pressure disturbance caused by the bend, make the fluid pressure change gradually stable, basically eliminate the high pressure area, and finally return to the fully developed turbulence state in the downstream.

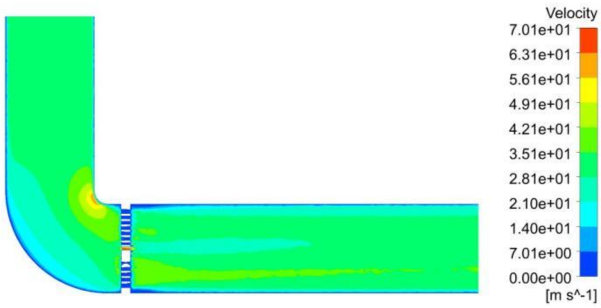

Figure 10: Pipe velocity contour with arc rectifier..

It can be seen from figure 10 that the circular arc rectifier can quickly eliminate the fluid velocity disturbance caused by the elbow, and finally present a fully developed initial state in the downstream. Since there is a solid area at the junction of the rectifier blades that chokes the flow, the velocity distribution near the center of the rectifier fluctuates slightly and there is a low velocity area.

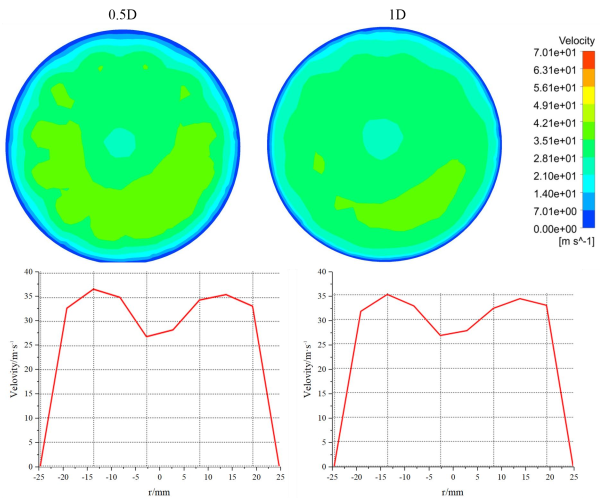

Figure 11: Velocity contour map of pipeline section with arc rectifier.

It can be seen from the schematic diagram of the flow velocity distribution of each section downstream of the elbow with the arc rectifier in figure 11 that after the fluid passes through the arc rectifier, the velocity curve at the 0.5D and 1D sections is not smooth enough, and the velocity is unevenly distributed; There is a part of the flow trough in the central area of the velocity distribution curve. This is because the central area of the arc rectifier is the confluence of the blades, which creates a solid area. When the fluid flows through the arc rectifier, this solid area produces an impact on the fluid. The blocking effect makes the flow velocity relatively slow. From 0.5D, various energy exchanges occur due to the interaction between the fluids, the jet section gradually increases, and the low-velocity region disappears. Finally, it has basically and fully developed when it reaches 1D. The flow curves are the same. At the 1D cross-section, the main flow area can be basically close to the fully developed velocity distribution, but there is still a slight deviation from the fully developed state in the area close to the tube wall. Part of the rectification effect is not very well.

Both rectifiers have a good rectification effect on the flow field, and the rectification effect on the fluid pressure field is basically the same. After rectification, the fluid pressure distribution is uniform. The rectification effect of the honeycomb rectifier on the fluid velocity field is more significant, and it can be seen that the velocity of the fluid is basically uniform after the distance from the rectifier 1D.

4. Conclusion

This paper analyzes the working principle and structural composition of the turbine flowmeter, and analyzes the flow field after the rectifier rectifier based on CFD simulation. The following conclusions are obtained: 1. The numerical simulation method can reflect the internal flow field information of the turbine flowmeter to a certain extent. , to provide auxiliary means for the structural improvement of the flowmeter, and the flow field uniformity index can effectively evaluate the rectification effect. 2. The distance of 1D reserved between the rectifier and the turbine, the flow velocity distribution is basically uniform, which meets the measurement requirements. 3. The diameter of the rectifier is consistent with the diameter of the impeller, which can improve the measurement accuracy and reduce the pressure loss. 4. The rectification effect of the cellular rectifier is better than that of the arc rectifier.

References

[1] Sun B, Chen S, Liu Q, et al. Review of sewage flow measuring instruments. Ain Shams Engineering Journal, ISSN: 20904479, 2020.

[2] Maha A. Nour, Muhammad M. Hussain. A Review of the Real-Time Monitoring of Fluid-Properties in Tubular Architectures for Industrial Applications. Sensors, 20(14): 3907, 2020.

[3] Guo S, Yang Z, Zhu Y, et al. Analysis of blade structure impact on turbine flow sensor performance. Flow Measurement and Instrumentation, 81: 102011, 2021.

[4] Blagojevič B, Širok, B, Bizjan B. Novel methodology for turbine gas meters error curve modelling across a wide range of operating parameters.tm-Technisches Messen, 88(11): 702-713, 2021.

[5] Guo S N, Yang Z H, Wang F, et al. Optimal design of wide viscosity range turbine flow sensor based on flow field analysis. Flow Measurement and Instrumentation, doi:10.1016/j.flowmeasinst.2021.101909.

[6] Guo S N, Yang Z H, Zhu Y, et al. Analysis of blade structure impact on turbine flow sensor performance. Flow Measurement and Instrumentation, 81: 101909, 2021.

[7] Chen Y , Ding W Z, Bian R.Performance studies on cavitation-resistance turbine flow sensor based on experiment and CFD simulation. Flow Measurement and Instrumentation,79:101918, 2021.

[8] Alaeddin M A, Hashemabadi S H. Study on the Effect of Different Operational Parameters on Ultrasonic Cross-Correlation Flowmeter Performance Using CFD Simulation.Journal of Petroleum Research,31(119): 39-49, 2021.

[9] Yao S, Su B, Yang Z L, et al. Influence of Upstream Elbow Pipe on Accuracy of Ultrasonic Flowmeter and Design of Rectification. Chinese Journal of Scientific Instrument, 43(05):102-109, 2022.

[10] Shao J C，Yan W W，Lin J D, et al. Structure optimization design of rear deflector for gas turbine flowmeter. Chinese Journal of Scientific Instrument, 43(01):46-53, 2022.

[11] Wang Z W，Yan W W，Xiao Y G, et al. Study on performance optimization of gas turbine flowmeter by numerical simulation and experimental measurement. Chinese Journal of Scientific Instrument, 43(01):28-34, 2022.

[12] Yuan Y Q, Li S Y, Zheng J. et al. CFD-Aided Investigation of Combined Flow Conditioners for Gas Ultrasonic Flow Meter. Journal of Shanghai Jiaotong University (Science), 28: 1-10, 2021.