DESIGN AND ANALYSIS OF FLOW RECTIFIER OF GAS TURBINE FLOWMETER

by

Zhi-Fei LI[a], Zheng DU[a], Kai ZHANG[b*], Dong-Sheng LI[b], Zhong-Di SU[b], and Wei-Wei YAN[b]

[a] Hangzhou Institute of Calibration and Testing for Quality and Technical Supervision, Hangzhou, China
[b] Department of Mechanics, China Jiliang University, Hangzhou, China

Short paper
DOI: 10.2298/TSCI1305504L

Three-dimensional computational model for a gas turbine flowmeter is proposed, and the finite volume based SIMPLEC method and k-ε turbulence model are used to obtain the detailed information of flow field in turbine flowmeter, such as velocity and pressure distribution. Comparison between numerical results and experimental data reveals a good agreement. A rectifier with little pressure loss is optimally designed and validated numerically and experimentally.

Key words: *pressure loss, numerical simulation, gas turbine flowmeter*

Introduction

The advent of CFD technology leads to a large number of studies on the complicated internal flow in flowmeters [1, 2]. The gas turbine flowmeters are the most commonly used components in gas piping line due to their stability and good linearity and play important roles in the gas piping management, being key elements in the estimation of piping line consumption. Along with the in-depth development of China's west-east gas transmission project, the gas turbine flowmeters are widely used in gas piping line to measure the flow-rate with high-accuracy. The wide use of turbine flowmeters can cause the increase of pressure loss, which simultaneously increases the energy consumption at delivery port. It is of great significance to decrease the pressure loss in turbine flowmeters to reduce energy consumption.

Based on the original model shown in fig. 1(a), two different kinds of fluid rectifiers were created to find which one can produce lower pressure loss in the gas turbine flowmeters. The key component of the gas turbine flowmeters is the flow sensor, which has moving parts that make the numerical simulation more complicated, here multiple reference frame (MRF) method [3, 4] was used to simulate the rotation of blades of turbine flowmeter. However, MRF method is not suitable for solving the stronger interaction between moving zone and stationary zone. Thus, the sliding mesh, which provides better accuracy for the simulation of moving parts, was applied to get the internal flow in flowmeters [5-7].

The impetus of this paper is to study the turbine flowmeter's pressure loss with two different kinds of fluid rectifiers. The sliding mesh and in-house codes were used to calculate the internal flow field and get the results.

* Corresponding author; e-mail: zkzb3026@gmail.com

Computational model

The flow filed in gas turbine flowmeter was numerically studied by the finite volume method. The flow was assumed to be compressible and steady. The present simulation adopted the following boundary conditions: nominal diameter 80 mm, flux scope 13~250 m^3/h, temperature 20 °C, and outlet pressure 101325 Pa. The length of straight pipe located at the inlet was 10 *D* and the length at the outlet was 5 *D*, where *D* is the diameter of thenpipe. The CAE software of Solidworks was used to create the 3-D geometric models, which were illustrated in figs. 1(b) and 1(c), respectively. Commercial software of GAMBIT was used to generate the mesh as shown in fig. 1(d). The number of elements of mesh was 200 million. Considering the existence of rotary part (impeller) in the meter, the sliding mesh was created carefully. The computational domain was divided into two parts: moving and stationary zone.

Original model (a) Model-A (b) Model-B(c) Numerical mesh(d)

Figure 1. Computational model of turbine flowmeter

For the steady and compressible flow, the dimensional governing equations can be written as:

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\mathrm{D}u_i}{\mathrm{D}t} = \rho F_i - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\mu}{3}\frac{\partial}{\partial x_i}\left(\frac{\partial u_k}{\partial x_k}\right) \tag{2}$$

$$\rho \frac{\mathrm{D}k}{\mathrm{D}t} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \tag{3}$$

$$\rho \frac{\mathrm{D}\varepsilon}{\mathrm{D}t} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 E\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} \tag{4}$$

$$G_k = \mu_t \frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{5}$$

Here

$$C_1 = \max\left(0.43, \frac{\eta}{\eta+5}\right),\ C_2 = 1.9,\ \mu_t = \frac{\rho C_\mu k^2}{\varepsilon},\ \eta = \frac{\sqrt{2E_{ij}E_{ij}}\,k}{\varepsilon}, \text{ and } E_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Results and discussion

With numerical simulation, the flow field with flow rate range from 13 to 250 m^3/h in the gas turbine flow meter was obtained. The impeller maintains a steady speed when the flow field is stable. Figure 2 shows the velocity magnitude of impeller at Qv = 13 m^3/h, 100 m^3/h, and Qv = 250 m^3/h, respectively, and it can be found that the largest flow velocity occurred on the outer edge of impeller, and the velocity distributions on the surface of impeller are very similar under these three different flow-rates.

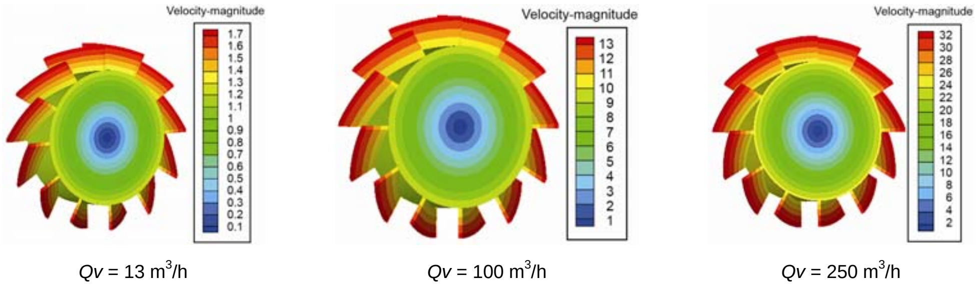

Figure 2. The velocity magnitude on the surface of the impeller *(for color image see journal web site)*

Figure 3 shows the contour of static pressure in the gas turbine flowmeter. It can be noted that, from the inlet to the outlet, the pressure loss seems more obvious, while there is little gradient of static pressure at the straight part of inlet and outlet at small or large flowrate. The static pressure changed a lot around the upstream of fluid rectifier and impeller, which means that pressure loss mainly caused by the complicated and rotational inner flow of blades and gaps between rectifier and pipe wall. Another different kind of upstream fluid rectifier was also assembled into the model to estimate which one is more effective to reduce the pressure loss.

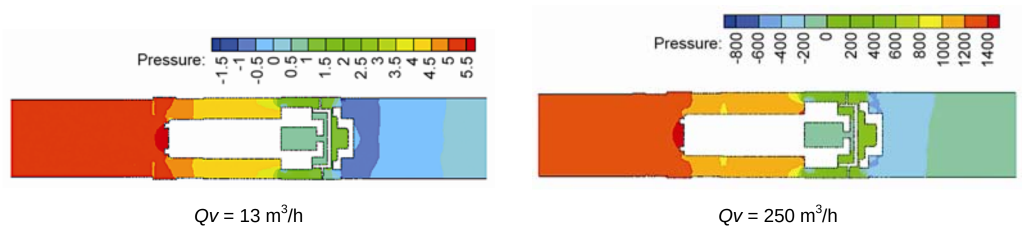

Figure 3. Contour of static pressure in the gas turbine flowmeter *(for color image see journal web site)*

The relationship between pressure loss and flowrate for model-A and model-B is illustrated in fig. 4. From figs. 4(a) and 4(b), it can be found that the pressure loss increased obviously with the growth of flow rate in both two models. Moreover, the pressure loss in

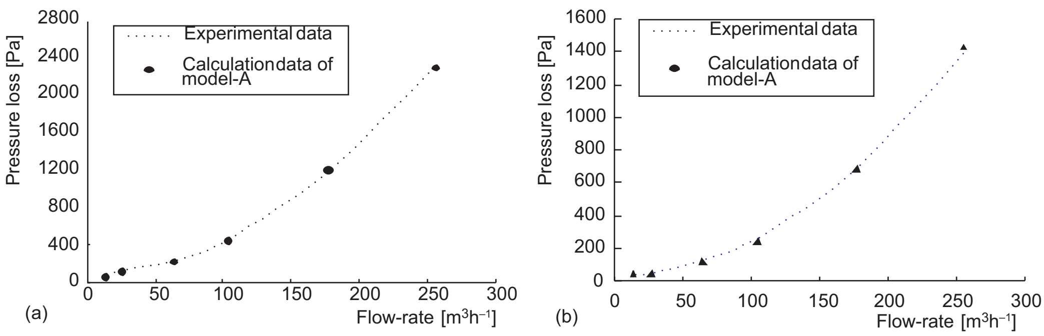

Figure 4. The relationship between flowrate and pressure loss

model-B is smaller than that in model-A, which suggests that the geometric model-B is superior to the model-A in reducing pressure loss in the gas turbine flowmeter. In addition, a lot of experimental work has been carried out, and it can be noted that the numerical results agree well with the experimental results as shown in fig. 4.

Conclusions

Pressure loss is one of the main technical parameters in gas turbine flowmeter. The computational results reveal that the present numerical method is reliable and effective in calculating the pressure loss in the flow sensor. Moreover, it is found that the geometric model-B is superior to model-A in reducing the pressure loss. Consequently, the present results are helpful for the design and optimization of gas turbine flowmeter.

Acknowledgment

The work was supported by Zhejiang Province innovation team project of Flow metering and online calibrating technology (2009R50024) and Qianjiang talent plan B of Zhejiang Province with Grant No. 2010R10014, National Natural Science Foundation of China with Grant Nos. 10902105.

References

[1] Zhao, X. D., *et al*., The Numerical Analysis of Variational Finite Element for Flow Field in the Rotor of Turbine Flowmeter and Turbine Performance Curve (Part one), *Proceedings,* 2th Flow Measurement Conference, London, 1988, Vol. 1, pp. 91-102
[2] Qin, H. X., *et al*., A Variational Finite Element Method with Variant Domain for Solving Fully 3-D Flow Field in Rotor of Turbine Meter, *Journal of Astronautic Metrology and Measurement*, *1* (1990), 6, pp. 50-54
[3] Du, G. S., *et al*., The Influence of Installation Conditions of Heat Meters on Interior Fluid Field and Flux Measurement Accuracy, *Journal of Hydrodynamics*, *18* (2006), 3, pp. 455-460
[4] Du, G. S., Liu, Z. G., Fluid Characteristic of Rotary Wing Heat Meter with Single-Channel, *Journal of Hydrodynamics*, *20* (2008), 1, pp. 101-107
[5] Sun, L. J., *et al*., Numerical Simulation of Turbine Flowmeters Three-Dimensional Flow Fields, *Proceedings,* 6th World Congress on Intelligent Control and Automation, Dalian, China, 2006, Vol. 1, pp. 5023-5027
[6] Wang, Z., Zhang, T., Computational Study of the Tangential Type Turbine Flowmeter, *Flow Measurement and Instrumentation*, *19* (2008), 3, pp. 233-239
[7] Lopez, G., *et al*., Modelling and Simulation of the Dynamic Performance of a Natural-Gas Turbine Flowmeter, *Applied Energy*, *83* (2006), 11, pp. 1222-1234

Paper submitted: January 18, 2013
Paper revised: April 26, 2013
Paper accepted: April 28, 2013