Numerical investigation of liquid flowmeter calibration device based on CFD

Yuan Zhuang¹, Ningning Zhang², Longbo Ma² and Yongmei Huang¹,³

¹ College of Metrology and Measurement Engineering, China Jiliang University, Hangzhou, Zhejiang, China
² Zhejiang Institute of Metrology and Science, Hangzhou, Zhejiang, China
³ E-mail: ymhuang@cjlu.edu.cn

Abstract. The emergence of new type flowmeters with higher accuracy has prompted us to study and improve the accuracy of standard calibration device. The structure of the calibration, especially the positions of the flowmeter, are important for its accuracy improvement. Based on Computational Fluid Dynamics (CFD), numerical simulation was used in this paper to investigate the flow velocity distribution in pipe at various distances from the diverting pipe to show the stability of the flow and to find the best location to install the test flowmeter. According to the simulation results, some suggestions are proposed for the device design in the end of the paper.

1. Introduction

The liquid flowmeter calibration device is the guarantee of the accurate transmission and the source of the accurate and reliable measurement. With the development of new technology, varieties of new high accuracy flowmeters have been developed. This requires a higher accuracy and reliability of the liquid flowmeter calibration device to meet the requirements of the flow value transmission.

The flowmeter performances are affected by their installation in different flow conditions [1]. The stability of liquid flow and the velocity distribution in the pipeline are the main factors that affect the accuracy of liquid flowmeter calibration. In the actual test, sufficient straight length is required to ensure the flowmeter accuracy. Roger C. Baker identified and measured the effect of manufacturing variation on the performance of high precision flowmeters installed in the calibration device [2]. Takashi Shimada identified the limits of accuracy of the same device (as mentioned in Ref. [2]) and the best obtainable uncertainty with this design of installation [3]. Kishor Borkar explored wafer cone flowmeter’s robustness in the presence of double 90° bend (out-of-plane) and gate valve as a source of upstream flow disturbance [4]. C. Ruppel, G. Bobovnik and A. Venugopal explored the measurement accuracy in different installation of ultrasonic flowmeter, Coriolis flowmeter and vortex flowmeter [5-7]. Above all, the structure and installation of the flowmeter are the key points for the application of calibration device.

Computational Fluid Dynamics (CFD) is helpful to the analysis of the installation effects. The information obtained by numerical simulation is much more extensive than in experimental investigations [8]. Zoheir Sabooohi proposed a numerical model for prediction of turbine flowmeter performance and a three dimensional steady state internal flow field of turbine flowmeter was obtained from the CFD simulations. The results were compared with experimental data and it was found that the numerical model results were reasonably accurate [9]. Aik Chong Lua determined the flow
characteristics of a target fluidic flowmeter by both experimental studies and numerical simulations. CFD was used in the numerical simulation and also to determine the best position to place the hot-film sensor for the detection of the maximum velocity oscillation signals in the flowmeter [10]. S. Shaaban proposed a novel flowmeter for liquid hydrogen and numerical simulation (CFD) was utilized in order to minimize the flowmeter’s loss coefficient and the required installation length [11]. Along with the progress of CFD, numerical simulation was found to be a useful tool for flow field analysis.

The main task of the paper was to discuss the flow field at various distances from the diverting pipe in order to obtain the appropriate positions for flowmeter installation. The following two cases are discussed:

a. The velocity distribution downstream of the diverting pipe with various opening degrees of the valve.

b. The velocity distribution downstream of the diverting pipe with various distances from the diverting pipe to the valve.

2. The calibrating device

This paper opted for the liquid flowmeter calibration device at Zhejiang Institute of Metrology and Science (ZJIM) which is a gravimetric system with a standing start and stop (where the flow is zero at the beginning and end of the run). The main flow coming from the head tank passes through the diverting pipe and will be divided into three selectable test lines whose pipe diameters ($D$) are 50mm, 80mm and 100mm respectively. The device has been shown in figure 1.

![Figure 1. Schematic diagram of the calibration device at ZJIM.](image)

The flowmeter was installed in the test pipeline, and the circulation system was applied so that the flow passed through the flowmeter and entered the weigh tank. While the water flowed into the weigh tank through the commutator, the flowmeter’s value and the measurement time were recorded. The output value of the flowmeter under testing was compared with the mass of the water collected in the weigh tank during the measurement time. With the certain mathematical calculation, the uncertainty of the flowmeter could be determined. According to figure 1, the straight pipe from the diverting point to valve and the length from the valve to the flowmeter were variable. Numerical simulation was used to predict the flow fields and understand the flow phenomena in these sections with variable length.
3. Model establishment and solution method

CFD is an analysis of systems which contain physical phenomena such as fluid flow and heat conduction by means of computer numerical calculation and image display. According to the essential step of CFD method, a model of fluid field was firstly established and divided fluid field into finite element for calculation. The FLUENT was used to do the solution and the results was post-processed and presented. The detailed steps and settings are as follows.

3.1. Model and meshing

The model of calibration device was established based on the actual experimental device. There were three test lines with different diameters in the device. The fluid field calculated in this paper was the pipe with diameter 100mm and in the middle of these three test lines as shown in figure 2. The fluid field was divided into grids after it had been extracted.

Tetrahedron mesh was selected for its good adaptability to complex structures and the element size was set as 9mm. Boundary layers were added at the walls in order to improve the accuracy of the solution and the first layer thickness was 2mm and the growth rate was 1.2. There were 2570881 nodes and 1606960 elements and the average quality of mesh was 0.78. The mesh of inlet was shown in figure 3.

3.2. Solution settings and boundary condition

Solution methods and boundary conditions were set according to the actual experimental environment. Transient was selected since the flow velocity was unsteady and changed with time as shown in figure 4 (the coordinate of x, y, z has been shown in figure 2). The inlet velocity of flow was set to 3m/s and the velocity of flow in the test section could be calculated as 6.9m/s according to the diameter ratio of the inlet pipe to the test section.

Turbulence is a complex and nonlinear flow model. RNG K-epsilon model was set as the Viscous Model since it had a better performance in dealing with high strain rate and flow with larger
streamline and the calculation results were easier to converge comparing with Standard K-epsilon model. The control equations of the turbulence model could be obtained by combining the continuity equation and the N-S equation as shown in equation (1) - (3).

The continuity equation:

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

The N-S equation:

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial u_i}{\partial x_j} \right] - \rho \nu \frac{\partial u_i^2}{\partial x_j} + S_i \]  

RNG K-epsilon model:

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu \frac{\partial k}{\partial x_j} \right] + G_k - \beta \nu \]  

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \alpha_\varepsilon \mu \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon} \frac{\varepsilon}{k} G_k - C_{\varepsilon} \nu \]  

Among them, \( \mu_{eff} = \mu + \mu_t \), \( \mu_t = \beta C_\mu \frac{k^2}{\varepsilon} \), \( C_\mu = 0.0845 \), \( \alpha_k = \alpha_\varepsilon = 1.39 \), \( C_{\varepsilon} = C_{\varepsilon} = \frac{\nu (1-\eta/\eta_0)}{1+\beta \nu} \), \( C_{1\varepsilon} = 1.42 \), \( C_{2\varepsilon} = 1.68 \), \( \eta = 2(E_{ij} * E_{ij})^{1/2} \frac{k}{\varepsilon} \), \( E_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \), \( \eta_0 = 4.377 \), \( \beta = 0.012 \).

The velocity-inlet and pressure-outlet were applied as the boundary conditions. SIMPLEC algorithm was applied to the calculation of flow for its good efficiency and fast convergence. Least Squares Cell Based was selected as Gradient to adapt to the polyhedral grid. Second Order was selected for Pressure and Second Order Upwind were applied to Momentum, Turbulent Kinetic Energy, Turbulent Dissipation Rate for higher calculation precision. Settings are shown in figure 5.

**Figure 5.** Settings of solution methods in Fluent.

3.3. Post-processing

Velocity distributions were created and displayed at different positions downstream of the diverting pipe and valve. The results have been displayed in chapter 4.

4. Results and discussion

Turbulent appeared when the flow passed through the diverting pipe and valve and made the velocity distribution quite different from the fully-developed flow. The difference was presented clearly through two contour plots of velocity in figure 6.
Figure 6. Contour plots in the pipeline.

Figure 7. The velocity distribution when the valve was fully-opened.

Figure 8. The velocity distribution when the valve was turned to 45°.

In order to get the required length of straight pipe section, the velocity distributions at various distances from the diverting pipe were presented according to the two cases described in the previous introduction.
In the case a, the flow changed with the varying of opening degree of the valve. The valve was set to fully-opened and turned to an angle of 45° respectively. The velocity distribution at 5D, 8D, 10D from the diverting pipe had been shown in figure 7 and 8. The distance between the diverting pipe and the valve was kept as 2D in this case.

Compared figure 7, 8. When the valve was fully-opened, it could be seemed as a straight pipe. So the flow was only influenced by the diverting pipe who caused a distorted velocity profile at 5D and the flow could almost reach to fully developed at 10D because the velocity curve was symmetrical and the maximum velocity appeared close to the center core instead of the edge according to figure 7. As the valve turned to 45°, however, the velocity distribution at 5D shown in figure 8(a) was distorted more significantly compared to figure 7(a). The velocity at the center was still smaller than the edge at 10D and longer straight pipe was needed to reach a fully developed flow. Therefore, it can be said that the valve which is not fully opened will enhance the flow disturbance in pipe line and longer straight length is required to ensure the flow stability.

Figure 9. The velocity distribution when $L=2D$.

Figure 10. The velocity distribution when $L=4D$.

Figure 11. The velocity distribution when $L=6D$. 
(2) In the case b, the flow changed with the varying of the valve’s position. The distance from the diverting pipe to the valve \((L)\) was 2\(D\), 4\(D\) and 6\(D\) respectively and the corresponding velocity distributions at 8\(D\), 12\(D\) and 15\(D\) from the diverting pipe had been shown in figure 9-11. The opening degree of the valve was kept as 45° in this case.

Compared figures 8, 9, 10. A higher peak velocity was obtained at 8\(D\) and the flow recovered more slowly with the increasing of the distance between the diverting pipe and the valve. The difference between the maximum velocity and the minimum velocity at 15\(D\) also increased according to figure 9(c.), 10(c.) and 11(c.) which meant worse-distribution of the velocity distribution was obtained at the same positions from the diverting pipe. Therefore, for the limited length of installation space, the disturbance flow in the upstream of flowmeter can be reduced in the way of shorten the distance from the diverting pipe to the valve as far as possible.

5. **Conclusions**

In this paper, based on the CFD method, numerical simulation was carried out to analyse the flow field in the flow calibration device. The diverting pipe and valve installed before the test section have great influence on the stability and velocity distribution of fluid flow. The main significance of this study is to provide guidance for flowmeter installation in the calibration device. According to the results and discussion in chapter 4, in order to reduce the influence of flow disturbance, the length of the sufficient straight pipe should be at least 15\(D\) from the diverting pipe. Besides, the distance from the diverting pipe to valve can be reduced properly for better measurement accuracy. Further experiments and numerical simulation can be carried out in the future research to find the optimum structure and improve the accuracy of the calibration device.

**References**

[1] Bobovik G and Kutin J 2013 Numerical analysis of installation effects in Coriolis flowmeters: A case study of a short straight tube full-bore design *Flow Measurement and Instrumentation* 34 142-150

[2] Baker R C and Wang T 2006 Observations on the design and development of a water flow rig related to calibration in the manufacturing process *Flow Measurement and Instrumentation* 17 171-178

[3] Takashi Shimada and Dharshanie V Mahadeva 2010 Further investigation into a water flow rig related to calibration *Flow Measurement and Instrumentation* 21 462-475

[4] Kishor Borkar and A Venugopal 2013 Pressure measurement technique and installation effects on the performance of wafer cone design *Flow Measurement and Instrumentation* 30 52-59

[5] C Ruppel and F Peters 2004 Effects of upstream installations on the reading of an ultrasonic flowmeter *Flow Measurement and Instrumentation* 15 167-177

[6] G. Bobovnik and J. Kutin 2015 Numerical analysis of installation effects in Coriolis flowmeters: Single and twin tube configurations *Flow Measurement and Instrumentation* 44 71-78

[7] A Venugopal and Amit Agrawal 2012 Frequency detection in vortex flowmeter for low Reynolds number using piezoelectric sensor and installation effects *Sensors and Actuators A: Physical* 184 78-85

[8] A Hilgenstock and R Ernst 1996 Analysis of installation effects by means of computational fluid dynamics - CFD vs experiments *Flow Measurement and Instrumentation* 7 161-171

[9] Zoheir Saboohi and Shahrok Sorkhhah 2015 Developing a model for prediction of helical turbine flowmeter performance using CFD *Flow Measurement and Instrumentation* 42 47-57

[10] Lua Aik Chong and Zheng Zhenheng 2003 Numerical simulations and experimental studies on a target fluidic flowmeter *Flow Measurement and Instrumentation* 14 43-49

[11] S Shaaban 2017 Design and optimization of a novel flowmeter for liquid hydrogen *International Journal of Hydrogen Energy* 42 14621-14632