The measuring system for air mass flow rate determination - difficult measuring conditions - case study

A Tomaszewski¹, T Przybyliński¹, M Lackowski¹, E Krzemieński², J Rogula³, G Romanik³, D Błoński³

¹ Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, PL-80-231 Gdańsk
² Zakład Produkcji Doświadczalnej Automatyki Co., Wysocka 46, PL-63-400 Ostrów Wielkopolski
³ Wrocław University of Science and Technology, Faculty of Mechanical and Power Engineering, Wybrzeże Wyspiańskiego 27, PL-50-370 Wrocław

E-mail: janusz.rogula@pwr.edu.pl

Abstract. Accurate gas flow measurement is required in many industrial applications. In typical circumstances the most reliable option is the use of single-hole orifice flowmeters, which characterize with the simplicity of the construction, lack of moving parts and high accuracy. However, such devices require long straight duct sections in order to perform the proper measurement. Sometimes on existing installations, where the space is limited, there is no possibility to fulfill these requirements. Such problem appeared in power plant in Opole. Multi-hole orifice flowmeters can be a reasonable solution in this particular case. Due to their enhanced resistance to flow disturbance they require shorter duct sections and could provide a correct measurement. The paper presents the results of the numerical analysis of air flow in 6-hole orifice flowmeter. The calculations were performed for the existing duct geometry from power plant in Opole. The orifice pressure drop was estimated for a wide range of Reynolds number. The analysis of the numerical results showed that multi-hole orifice flowmeters can be used in investigated situation. The results of experimental measurements performed in Opole power plant are also included in the paper.

1. Introduction
The subject of this research is an innovative flow measurement system which is also a ball valve. As a part of the research, a critical analysis of the latest measuring orifice solutions was carried out. The subject analysis focused on problems with the reliability of measurements performed in non-standard conditions, including primarily the situation of non-compliance with the normative lengths of straight sections upstream and downstream the measuring element. Such situation occurred in the power plant in Opole - the existing installation should measure the air flow at 8 bar pressure in the duct with diameter of D = 100 mm.

In accordance with ISO 5167-2 standard [1], after the 90° turn, an inlet section with a length of up to 40 D and 20 D on the discharge side should be used. This corresponds to 4 m and 2 m length on both sides.
of the orifice. On a real object, without serious interference, it is not possible to carry out the measurement correctly. The lengths of the mentioned sections, recommended by the standard ISO 5167-2, depend on the orifice narrowing - the smaller the contraction coefficient, the greater the resistance to flow disturbance and, consequently, the shorter the length of the required sections on the inflow and outflow side. On the other hand, the smaller the contraction coefficient, the greater the pressure drop on the measuring element and the greater the required power demand in the pumping system. The contraction coefficient had to be chosen accordingly to meet both requirements. Unfortunately, no reasonable value of narrowing coefficient for the single-hole orifice allowed correct measurement in the analyzed case.

Based on the literature analysis, it was noticed that multi-hole orifices usually have greater resistance to the non-uniform flow of the medium in the pipeline, which could be used in the described situation. However, such solution would constitute an experimental use not included in any applicable standard. For the purposes of further research, it was considered that the contraction coefficient of the multi-hole orifice should be around $\beta \approx 0.5$. The proposed orifice geometry has been a subject of further calculations. Using computational fluid mechanics CFD, it was decided to study the flow structures around the measuring orifice as well as the pressure drop and mass flow dependency.

There are many papers available in scientific literature regarding numerical simulation of fluid flow in multi-hole orifice flow meters. Below few of them are described.

One of the first such simulations [2] concerned 2D axisymmetric modelling of flow through single-hole orifice. The results were compared with experiment, showing good agreement with respect to pressure drop on the orifice. Various discretization schemes and turbulence model parameters were tested by the authors and best results deviated only 0.2 % from the measured pressure drop. More recently, similar computational analysis was presented in [3] for water flow. The results discussed in [3] focused on turbulent energy distributions but also calculated axial velocity and centerline pressure profiles were shown. The accuracy of this simulations, in terms of discharge coefficient, reached 0.8 % for the pipe flow Reynolds number $Re = 5760$.

With regard to multi-holed orifices, CFD simulation results can be found e.g. in [4-9]. The simulations for seven variants of a 9-hole orifice were published in [5]. All of the orifices had the same diameter ratio $\beta = 0.5$, a central hole and eight peripheral holes arranged on a circle. They differed in relative size of the central hole to the peripheral ones and in the diameter of the circle on which the latter were distributed. On the basis of reference flow measurement and calculations with a single-hole orifice, the CFD simulations accuracy was estimated at 7 %. The results shown in [5] demonstrated that the disturbance in the flow velocity field downstream of the multi-hole orifice disappears over shorter length of the pipeline than for a standard orifice. A region of separated flow for multi-hole orifice is also much smaller. From the analysis of the pressure field it is clear that the loss of pressure in the analyzed multi-hole orifice is smaller than in the standard one and the discharge coefficient is higher.

A series of CFD simulations of flow through a 4-hole orifice, aimed at pressure loss minimization, can be found in [6]. This aim was reached by optimization of the holes longitudinal profile, which were in the shape of convergent-divergent nozzles. The optimized orifice demonstrated lower values of velocities, exergy destruction, turbulence and pressure drop on the downstream side as well as exceptionally high discharge coefficient, comparable to the Venturi nozzle.

Simulations similar to the above mentioned can also be found in [4, 7] but they concern multi-slotted orifices (i.e. with rectangular holes) which were meant to wet gas flow measurement. It can be seen from velocity fields presented in [4] that flow disturbance downstream of the orifice decreases with increase of the slot aspect ratio (length to width ratio). In comparison, the equivalent standard one-hole orifice generates the highest flow disturbance, while the multi-circular-hole orifice is positioned in this regard between the standard and the multi-slotted orifices. Although the velocity fields calculated in [4] for multi-slotted and multi-holed orifices differ, the pressure drop due to their presence in the flow is almost
identical. It is, as already reported in [5, 6], significantly lower than in the case of an equivalent standard orifice.

Among works presenting CFD results there is also, [8] which discusses the calculation of air flow through a perforated plate. There is, among others, an analysis of the pressure loss coefficient dependence on the inclination (impact) angle of the flow through a perforated plate. For a plate for which the ratio of hole diameter to plate thickness is 0.3 and porosity equals to $\beta=0.4$, it has been found that the pressure loss is practically constant for inclination angles from 0° to approx. 30°. For thinner plates this range is even wider. Such results may suggest the increased resistance of multi-hole orifices to flow velocity disturbances.

In this paper, flow characteristics of six-hole orifice were investigated theoretically by computational fluid dynamics (CFD). Special attention was paid to determine how the disturbance of upcoming flow velocity field, as an effect of the channel geometry influences the orifice differential pressure. The motivation to this study is an innovative idea of a ball valve with such an orifice embedded in a way that allows (Fig. 2) for its easy exchange.

The impact of the flow disturbance introduced by the existing channel geometry, where the straight sections recommended by the standard were not maintained, on the measuring accuracy of the tested measuring orifice was analyzed.

2. Numerical model
The geometry of the investigated orifice is shown in Fig. 1. It has the contraction coefficient of $\beta = 0.5164$ and 6 hole situated symmetrically on a circle with 30 mm radius.

![Figure 1. The hole profile of the investigated orifice.](image)

As mentioned in the introduction, the existing installation in power plant in Opole do not allow using a required upstream and downstream sections recommended by the standards for single hole orifice. The geometry of the existing channel with mounted multi-hole orifice is presented in Fig. 2. The inlet section length is 8D while the outlet section has a length of 2D.
In order to investigate the pressure drop and mass flow dependency on mounted orifice, the numerical model was prepared. The 3D channel geometry, being a mapping of real channel, has been created and is presented in Fig. 3.
In order to perform the discretization process properly the computational domain has been divided into separate sections – blocs, which were later connected with mesh interfaces. The computational mesh was fine near the orifices and coarse near inlet and outlet. The boundary layer consisted of 15 inflation layers. The first layer, closest to the wall was situated in such distance to fulfill the condition of $Y^+ = 3$. The total number of elements was almost 10 mln.

The numerical simulation was performed using commercial CFD code to solve the steady state Reynolds Averaged Navier–Stocks Equation (RANS). The working fluid was dry air. Its compressibility was taken into account in the simulations. Therefore, ideal gas model was used in order to calculate the density of air and energy balance equation also had to be included. The turbulence model used in simulation was realizable $k–\varepsilon$, which according to Shaaban [12] should lead to proper and accurate results. The boundary condition on inlet was “mass flow inlet” with inlet temperature 15°C. The boundary condition on outlet was “pressure outlet” with the absolute value of air pressure set to 8 bar. The pressure–velocity coupling scheme was set to SIMPLE. The applied discretization schemes are of second order accuracy for pressure, density, momentum, turbulent kinetic energy, turbulent dissipation rate and energy.

3. Results
The calculations were performed for wide range of mass flow introduced in the inlet. After the solution was converged, the pressure was probed immediately upstream and downstream the orifice so that the pressure drop was determined. The characteristics of multi-hole orifice flow meter is presented in Fig. 4.

![Graph showing pressure drop and mass flow rate characteristics for the investigated orifice in working conditions.](image)

Figure 4. Pressure drop and mass flow rate characteristics for the investigated orifice in working conditions.

After the prototype of the valve was made, preparations for experimental measurements began. The orifice was placed on the existing installation at the Opole power plant, where the pressure drop at the orifice under test and the temperature and total pressure in front of the orifice housing were measured. The
pressure drop was measured on the six-hole orifice flow meter and then the volumetric flow rate of the medium was calculated from the dependency presented in Fig. 4. Obtained value of flow rate was changing rapidly – in Fig. 5 there is presented data for 1 hour of constant work of the installation.

**Figure 5.** Flow rate change.

#### 4. Conclusion
The solution for fluid mass flow determination in non-standard conditions were proposed. The six-hole orifice flow meter was mounted on existing installation where the space is limited and there was no possibility to place the inlet & outlet sections of proper length recommended by the standards. The numerical model of flow through existing channel geometry and chosen six-hole orifice flow meter was prepared. The calculations were performed for different mass flow introduced at the inlet. The mass flow and pressure drop dependency was investigated for working conditions. The obtained characteristics allowed to estimate the flow rate in existing installation in power plant in Opole.

#### References
[1] ISO 5167-2:2003, Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full – Part 2: Orifice plates *International Organization for Standardization (ISO)* (Geneva, Switzerland)
[2] Kumar P, Bing M W M 2011 A CFD study of low pressure wet gas metering using slotted orifice meters *Flow Measurement and Instrumentation* **22** 33
[3] Erdal A, Andersson H I 1997 Numerical aspects of flow computation through orifices *Flow Measurement and Instrumentation* **8** (1) 27
[4] Shah M S, Joshi J B, Kalsi A S, Prasad C S R, Shukla D S 2012 Analysis of flow through an orifice meter: CFD simulation *Chemical Engineering Science* **71** 300
[5] Singh V K, Tharakan T J 2015 Numerical simulations for multi-hole orifice flow meter *Flow Measurement and Instrumentation* **45** 375
[6] Shaaban S 2015 On the performance of perforated plate with optimized hole geometry *Flow Measurement and Instrumentation* **46** 44
[7] Geng Y, Zheng J, Shi T 2006 Study on the metering characteristics of a slotted orifice for wet gas flow *Flow Measurement and Instrumentation* **17** 123

[8] Guo B Y, Hou Q F, Yu A B , Li L F, Guo J 2013 Numerical modelling of the gas flow through perforated plates *Chemical Engineering Research and Design* **91** 403

[9] Bae Y, Kim Y I 2016 Numerical modelling of an isotropic drag for a perforated plate with cylindrical holes *Chemical Engineering Science* **149** 78