Numerical investigation of wet gas flow in Venturi meter

Denghui He, Bofeng Bai*

State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China

A R T I C L E   I N F O
Available online 8 August 2012
Keywords:
Wet gas
Over-reading
Turbulence model
Discrete phase model
Liquid jet
Venturi meter

A B S T R A C T
Study of the Venturi meter over-reading in wet gas is of considerable importance for the wet gas metering. Although the impacts of different parameters (e.g., liquid fraction, pressure and gas flow rate) on the over-reading have been widely investigated, the underlying mechanism on how these parameters act on the over-reading is still not fully understood. In this investigation, five types of turbulence models, including the standard k-ε model, the RNG k-ε model, the realizable k-ε model, and the Reynolds stress model were examined. It was found that the standard k-ε model was in better agreement with the experimental data. From the simulations, how and why the over-reading produced was explained. Then the liquid phase distributions and its impact on the velocity field and the pressure profiles were discussed. The results indicated that the liquid accumulated in the convergent section of the Venturi tube, where an annular liquid jet was formed. The static pressure in the throat declined along the throat, which made the static pressure in the throat unstable. To reduce their adverse effects on the over-reading of the wet gas flow, it was suggested that the classical Venturi tube should extend the length of the throat and decrease the convergent angle. This study gained a more comprehensive understanding of Venturi meter wet gas over-reading and provided a reference for the design of a wet gas Venturi meter prototype.

© 2012 Elsevier Ltd. All rights reserved.

1. Introduction

Wet gas flow measurement is becoming increasingly important to the production of natural gas [1,2]. The wet gas meter designs employ mostly the Differential Pressure (DP) meter technology [3–8], especially the Venturi tube due to its proven advantages, including safety, economy, convenience and clear physical interpretation. Although the general understanding of the Venturi tube performance in wet gas is widely accepted [3], little is known about the internal interactions of the wet gas flow in a Venturi meter.

When the Venturi meter is used in the wet gas flow, the DP with wet gas flow is usually larger than it would be if there was no liquid present with the gas. This usually causes a positive error of the gas flow rate prediction of the DP meter. Therefore, it is said that the meter is “over-reading” (thereafter OR). The OR is the ratio of the apparent gas mass flow, \( m_{g,\text{Apparent}} \), to the gas mass flow rate \( m_g \).

\[
\text{OR} = \frac{m_{g,\text{Apparent}}}{m_g}
\]  

\[
\text{Apparent} = \frac{\pi D^2 C_d \sqrt{\frac{2 \rho_g}{\rho_l}} \sqrt{\frac{m_g}{\rho_l}} \Delta P_{tp}}{4 \sqrt{1 - \beta^4}}
\]

where, \( d \) and \( D \) are the throat diameter of the Venturi and the pipe diameter, respectively, \( \beta \) the diameter ratio \( (\beta = d/D) \), \( C_d \) the discharge coefficient, \( \epsilon \) the expansibility factor, \( \rho_g \) the gas density, and \( \Delta P_{tp} \) is the wet gas differential pressure.

Many investigators [6,9–12] reported that the OR of the Venturi meter is dependent on the Lockhart–Martinelli parameter (\( X_{LM} \) defined by Eq. (3)), the operating pressure \( (P) \), the gas densimetric Froude number \( (F_{rg}) \) defined by Eq. (4)) and the diameter ratio \( (\beta) \). It is generally accepted that the OR increases with the increase of \( X_{LM} \) or \( F_{rg} \) keeping other parameters constant and decreases with the increase of \( P \) and \( OR \) also decreases as \( \beta \) increases. In addition, several researchers [13,14] found that the Venturi OR was correlated with the liquid phase property and the pipe diameter.

\[
X_{LM} = \frac{m_l}{m_g} \sqrt{\frac{\rho_g}{\rho_l}}
\]  

\[
F_{rg} = \frac{U_{sg}}{\sqrt{gD}} \sqrt{\frac{\rho_g}{\rho_l}}
\]  

\[
U_{sg} = \frac{m_g}{\rho_g} \frac{4}{\pi D^2}
\]

where, \( m_l \) is the liquid mass flow rate, \( \rho_l \) is the liquid density, and \( U_{sg} \) is the superficial gas velocity.

In the last few years, Computational Fluid Dynamics (CFD) has been applied increasingly in wet gas metering and various models have been used. Reader-Harris et al. [15] examined wet gas flow through Venturi tubes, in which the Euler–Euler multiphase model was employed. They noted that it was possible to model...
wet gas flow through Venturi tubes and provided good tendency with the experimental over-reading data. Xu et al. [16] presented a simulation method based on the Discrete Phase Model (DPM) to predict the OR characteristics of the Venturi tube. In their study, the Renormalization Group (RNG) $k$-$\varepsilon$ model was used. They reported that the maximal relative error of OR was 5.14%, and the average relative error was less than 2.8%. The effects of installation, tapping length and different types of gases on Venturi tubes and the derivation of the discharge coefficient were investigated in [17]. It was found that the standard $k$-$\varepsilon$ model appeared in better agreement with the test data than the Reynolds Stress Model (RSM). Moreover, in several investigations [16,18,19], the RNG $k$-$\varepsilon$ model and DPM were used to investigate wet gas flow in a V-cone meter. Comparisons between the simulations and the experiments suggested that the CFD model worked well on the OR prediction. Although the single-phase fluid flow in the Venturi tube is well known to us, the wet gas flow is still under exploring. The objective of this research is to simulate the flow field characteristic of the wet gas flow in Venturi meter by proposing a numerical model. Five turbulence models were compared. And the SKE model was selected. Then the liquid phase concentration, velocity and pressure distributions were investigated. On the basis of the results, some advice on the improvement of the classical Venturi tube were provided to get a more preferable performance in wet gas flow.

2. Computational details

2.1. Geometry and experiment

The experiments to validate the simulations are from a report made by the NEL (National Engineering Laboratory) in the UK [11]. The Venturi tube is shown in Fig. 1 [20]. The diameter ratio ($\beta$) of the Venturi tube is 0.75 and the pipe diameter (D) is 100 mm, the convergent and divergent angles are 21° and 7°, respectively. The high pressure tappings (upstream pressure tappings) are 50 mm away from the entrance of the conical convergent, the low pressure tappings (throat pressure tappings) are in the middle of the cylindrical throat.

The geometry consists of three parts, i.e. the upstream pipe, the Venturi tube and the downstream pipe, as the 3D view of the modeled computational flow domain shown in Fig. 2. The upstream straight length is three times the pipe diameter from the entrance of the conical convergent and downstream straight length is 10 diameters downstream from the end of the cone, which enables the flow to fully develop and the pressure building to finish. The flow domain was meshed with structured hexahedral meshes and the boundary layer meshing scheme was used for grid generation in the region proximate to the wall. Moreover, the mesh size of the throat was kept fine enough to achieve better convergence and greater accuracy. The grid independency was tested using computational grids among 300 000 and 1200 000 cells. The computational grid of approximately 660 000 cells was selected here because of its economic computation and perfect prediction.

Table 1

<table>
<thead>
<tr>
<th>$\beta$</th>
<th>Pressure (MPa, gauge)</th>
<th>Fr$_g$</th>
<th>$X_{LM}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.75</td>
<td>1.6</td>
<td>1.5, 2.5, 3.5</td>
<td>0–0.30</td>
</tr>
<tr>
<td>3.1</td>
<td>1.5, 2.5, 3.5, 4.5</td>
<td>0–0.30</td>
<td></td>
</tr>
<tr>
<td>6.1</td>
<td>1.5, 2.5, 3.5, 4.5</td>
<td>0–0.30</td>
<td></td>
</tr>
</tbody>
</table>

2.2. Mathematical model

2.2.1. Turbulence model

The commercial CFD software, FLUENT 6.3, was used here. The continuum gas phase (nitrogen) was predicted under steady-state conditions. Five types of turbulence models [20], i.e., the SKE, the RNG, the Realizable $k$-$\varepsilon$ model (hereafter RKE), the Standard $k$-$\varepsilon$ model (hereafter KWM) and the RSM were compared in this study. The SKE, RNG, and RKE models have similar forms, with transport equations for $k$ and $\varepsilon$. The major differences in the three models are as follows: the method of calculating turbulent viscosity, the turbulent Prandtl numbers governing the turbulent diffusion of $k$ and $\varepsilon$ and the generation and destruction of the turbulence in the $\varepsilon$ equation.

The KWM model contains the modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. This model is in close agreement with measurements for far wakes,
mixing layers, and plane, round, and radial jets, and is thus applicable to wall-bounded flows and free shear flows [21].

The RSM [22–24] accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a rigorous manner, and it has great potential to give accurate predictions for complex flows, such as cyclone flows, highly swirling flows in combustors, rotating flow passages, and the stress-induced secondary flows in ducts [21].

2.2.2. Multiphase model

In the Annular-Mist flow, the liquid consists of two types: the droplet and very thin liquid film. In the present simulation, the maximum volume fraction of the liquid is 8.23%. Thus the DPM model is fully capable of simulating the conditions (the DPM model usually used for the liquid fractions is less than 10% [21]). In addition, to simulate the thin liquid film, the wall-film model was also used as the boundary condition of the wall. The equations of the motion for droplets can be written as

\[ \frac{d\mathbf{u}_d}{dt} = F_d(u_d - u_l) + \frac{g_d}{\rho_d} \left( \frac{\rho_l - \rho_d}{\rho_l} \right) + F_s \]  (6)

\[ F_D = \frac{18\mu_d C_D Re}{\rho_d d_l^2} \]  (7)

where \( u_d, u_l \) are the gas and liquid velocity, respectively, \( \rho_d \) the gas density, \( \rho_l \) the liquid density, \( g_d \) the gravitational acceleration, \( F_d(u_d - u_l) \) the drag force per unit droplet mass, and \( F_D \) is determined by Eq. (7), \( F_s \) is the Saffman lift force due to the shear between phases, \( \mu_d \) is the gas molecular viscosity, \( d_l \) the liquid droplet diameter, and \( Re \) is the relative Reynolds number defined as Eq. (8), \( C_D \) is the drag coefficient [25].

\[ Re = \frac{\rho_d d_l |u_d - u_l|}{\mu_d} \]  (8)

\[ C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \]  (9)

where \( a_1, a_2, a_3 \) are empirical constants for smooth spherical droplets over several ranges of droplet Reynolds number.

Table 2

<table>
<thead>
<tr>
<th>Diameter range (µm)</th>
<th>Mass fraction in range</th>
<th>Diameter, ( d_0 ) (µm)</th>
<th>( Y_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0–50</td>
<td>0.05</td>
<td>50</td>
<td>0.95</td>
</tr>
<tr>
<td>50–100</td>
<td>0.05</td>
<td>100</td>
<td>0.90</td>
</tr>
<tr>
<td>100–150</td>
<td>0.15</td>
<td>150</td>
<td>0.75</td>
</tr>
<tr>
<td>150–200</td>
<td>0.20</td>
<td>200</td>
<td>0.55</td>
</tr>
<tr>
<td>200–250</td>
<td>0.20</td>
<td>250</td>
<td>0.35</td>
</tr>
<tr>
<td>250–300</td>
<td>0.15</td>
<td>300</td>
<td>0.20</td>
</tr>
<tr>
<td>300–350</td>
<td>0.10</td>
<td>350</td>
<td>0.10</td>
</tr>
<tr>
<td>350–400</td>
<td>0.05</td>
<td>400</td>
<td>0.05</td>
</tr>
<tr>
<td>400–500</td>
<td>0.05</td>
<td>500</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Furthermore, the stochastic tracking (random walk) model was used to predict the dispersion of droplets due to turbulence in the gas phase.

2.3. Numerical procedure

The governing transport equations were discretized with a finite-volume approach. The second-order upwind discretization scheme was used for the pressure equation and the third-order QUICK scheme was adopted for other terms.

The mass flow inlet boundary condition was used to define the gas flow rate at the flow inlet and pressure outlet boundary condition was adopted at the end of the pipeline. The turbulence intensity at the inlet and outlet was dependent on the empirical correlation for fully-developed duct flows. The boundary condition of the wall employed the wall-film model. The temperature was set as 291.15 K.

The liquid (liquid kerosene) was injected from the surface at the inlet. The distribution of the droplet sizes employed the Rosin–Rammell type, the mass fraction \( Y_d \) of the droplets diameter greater than \( d_0 \) was given by

\[ Y_d = \exp\left( -\left( \frac{d_0}{\bar{d}} \right)^n \right) \]  (10)

where \( \bar{d} \) is the droplet mean diameter, \( n \) is the spread parameter. Table 2 shows the distribution of the liquid droplet diameter in present simulation. The minimum and maximum diameters of the droplet are 0.05 mm and 0.5 mm, respectively, and the mean diameter is 0.248 mm [26].

To increase the calculation efficiency, the continuous gas phase flow field was obtained firstly and the liquid phase simulation was then carried out based on the converged solution of the gas flow. The convergence criteria were assumed to be met when the iteration residuals were reduced by \( 10^{-6} \).

3. Comparison of turbulence models

According to ISO 5167-4:2003 [20] the discharge coefficient \( C_d \) of the classical Venturi tube with a machined convergent section is approximately 1.000 when the throat Reynolds number, \( Re_t \), lies between \( 10^{6} \) and \( 2 \times 10^{6} \) and \( C_d = 1.010 \) with \( Re_t \) ranging from \( 2 \times 10^{6} \) to \( 4 \times 10^{6} \). Table 3 shows the discharge coefficient with different turbulence models compared with the values recommended by ISO 5167-4:2003. All turbulence models can predict the single phase gas flow accurately. There are little differences in relative deviations of all five models and the maximum deviation is less than 2.6%.

As can be seen from Fig. 4, the five models under predict the OR for low \( X_{IM} \) (< 0.05) and over predict for high \( X_{IM} \) at \( Fr_t = 1.5 \); whereas the simulations of the five models under predict the OR at \( Fr_t = 3.5 \). The SKE model gives slightly higher OR than the other four models. Fig. 5 shows the relative deviations of OR at \( Fr_t = 1.5 \) and \( Fr_t = 3.5 \) for different turbulence models. At \( Fr_t = 1.5 \), the KWM model gives the smallest relative deviation of OR, the deviation of the SKE model is the largest, but
it is still no more than 6.0%. When Frg = 3.5, the SKE model predicts the wet gas best among the five models, while the deviation of the KWM model is up to 8.5%. Hence, compared with the other four models, the SKE model can predict the wet gas better. And the relative deviation of the OR is with ±6.0%.

From the above discussions, the SKE model is better than other four models in the wet gas simulation. In addition, the convergence of the other four models is more difficult than the SKE model, especially the RSM model. Hence the SKE model is finally selected.

4. Results and discussion

4.1. Comparisons between simulations and test results

The comparisons between the simulations and the test results [11] under different pressure are shown in Fig. 6. The simulations agree well with the experiments. The OR is closely related to the XLM and increases with it for other parameters held constant. Moreover, an increase in the pressure leads to a reduction in OR.

As shown in Fig. 7, the OR does not vary significantly with the Frg. On the one hand, the effect of the Frg on the OR is closely related
with the wet gas flow pattern [10]. In our investigation, the flow pattern appeared as Annular-Mist flow in which most of the liquid moved close to the gas velocity as small droplets. Under such conditions, increasing gas velocity has little effect on the OR. On the other hand, the droplet size distribution shown in Table 2 is the same for different Frg. However, the predicted OR varies with droplet size and are close to each other for different Frg under the equal droplet size [15].

The Relative deviations of predicted OR compared with the experimental data [11] are displayed in Fig. 8. The relative deviation of the OR is within $\pm 5.5\%$ at the 95% confidence level. The maximal relative deviation of OR is 6.38%, and the average relative deviation is less than 2.84%. The comparisons with the experiments show that the model and its solution approach are reasonable.

### 4.2. Liquid phase concentration distributions

The concentration of liquid is quite intense in the convergent and throat sections of the Venturi tube, for the droplets impacting on the wall of the convergent section and forming a liquid layer as shown in Fig. 9. This liquid layer then separates at the end of the convergent and forms an annular jet entering the throat, after which it continues to pass through the divergent without reattachment to the wall. A similar phenomenon was also reported by Reader-Harris et al. [15].

When the liquid fraction is low (e.g. $X_{LM} = 0.01$), the liquid jet is not noticeable and most of the liquid is carried by the gas in droplet form and disperses into the gas more homogeneously, as shown in Fig. 10(a). The friction pressure drop and the acceleration pressure drop are almost the same as that in the dry gas flow, so the presence of the liquid produces very low OR (Fig. 6). The liquid jet becomes obvious and lasts a much longer distance as the liquid fraction increases and the throat area occupied by the liquid phase increases. Thus, as shown in Fig. 11, the effective gas flow passage in the core of the throat is decreased when the $X_{LM}$ is from 0.01 to 0.3, which leads to the increase of the acceleration pressure drop. The OR increases correspondingly as shown in Fig. 6.

Under the conditions of fixed pressure and $X_{LM}$, the gas flow rate has little effect on the distribution of the liquid phase concentration as shown in Fig. 10(b). The liquid jet is not affected by the increasing Frg. Under the conditions of fixed Frg and $X_{LM}$, it is shown from Fig. 10(c) that the liquid annular jet is more obvious under lower pressure than that under higher pressure, which leads to greater OR as shown in Fig. 6.

### 4.3. Wall pressure profile distributions

When the $X_{LM}$ is low (e.g. $X_{LM} \leq 0.05$), the liquid has little effect on the wall pressure profile as shown in Fig. 12. Like the dry gas, the
5. Conclusions

In this investigation, the wet gas flow through a Venturi meter was examined with the Discrete Phase Model. The standard k-ε model agreed with the experimental data better and was employed. On the basis of the simulations, the liquid phase distributions and the pressure profiles and their impact on the over-reading of the wet gas Venturi meter were discussed. The strategies to reduce their adverse effect on the measurement were suggested. The main findings may be summarized as follows:

(1) The liquid accumulated in the convergent section of the Venturi tube and formed an annular liquid jet. The liquid jet was much more obvious under both greater liquid fraction and lower pressure, which led to greater over-reading.

(2) The static pressure in wet gas flow is unstable and declines much more along the throat than that in dry gas flow. The greater the XLM is, the faster the throat pressure declines.

The decline of the pressure is bad for the pressure measurement in the throat of the Venturi.

Acknowledgments

This work was financially supported by the National Nature Science Foundation of China for Creative Research Groups under Contract No. 51121092.

References


Larsen M, Howell J. Least-squares smoothing of direct-exchange areas in zonal analysis. Sandia National Labs., Albuquerque, NM (USA); Texas Univ., Austin (USA), Dept. of Mechanical Engineering; 1984.

Fig. 12. Wall static pressure profiles under 3.1 MPa gauge (Fg = 2.5, XLM = 0–0.3).