CFD analysis of flow through Venturi tube and its discharge coefficient

A Tukimin1*, M Zuber2 and K A Ahmad2

1Department of Mechanical and Manufacturing Engineering, Faculty of Engineering, Universiti Putra Malaysia, Malaysia
2Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia, Malaysia

*agustinnie@gmail.com

Abstract. Venturi tube plays a very important role in different fields of engineering. It has a number of industrial applications in which its design is an essential factor. Venturi tube used in gas measurement applications provides an accurate critical gas flow measurement. There is a need to design Venturi tube with an effective analytical tool or software. In this work, two parameters: pressure drop and velocity discharge nozzle were analyzed using Computational Fluid Dynamics (CFD). The results obtained were then analyzed for accurate determination of the Venturi tube’s discharge coefficient, Cd. It was found that there is less than 1% difference between the average values of the discharge coefficient obtained from the numerical analysis and experimental results.

1. Introduction

Due to the gaining popularity of Venturi tubes in the gas and wet gas measurement applications, there are increasing studies being developed to understand its behaviours in the application in order to fully benefit from its use. Earlier works have shown that the performance of Venturi tubes in gas is very different from that in water [1, 2]. As the understanding of Venturi tubes’ performance in dry gas is the priority before its performance in wet gas can be analysed, the National Engineering Laboratory (NEL) in the United Kingdom, supported by Elf Exploration, Jordan Kent Metering Systems/Seiko, Mobil North Sea, and Shell International, had conducted the experimental and numerical work to investigate the Venturi tubes’ discharge coefficient behaviour in dry gas [3]. The results of the work indicated that the discharge coefficient numerical models are very different between gas and water applications. In addition to the numerical models, further analysis also indicated the Venturi tube surface roughness could impact the discharge coefficient error by more than 5%.

In general, most of the Venturi tubes have to be calibrated either using dry gas or water for the discharge coefficient determination prior to being used in the actual application in order to improve their measurement accuracy. This paper presents the CFD analysis that was carried out to verify the flow through Venturi meter and the discharge coefficient determined by the experiment using dry gas at the high-pressure wet-gas test facility at National Engineering Laboratory (NEL) in Glasgow, United Kingdom. The objectives are to investigate the most effective models for numerical analysis for discharge coefficient determination and to develop faster and cheaper methods for discharge coefficient determination for practical application, especially for cases where big numbers of Venturi meters are involved. The Venturi tube used in this study is manufactured for the well testing loop on a well head platform for PETRONAS Carigali Sdn. Bhd. (PCSB) SK316 Field Development project.
The CFD analysis is the first step for subsequent analysis of the Venturi tube design for the wet gas application.

2. Experimental setup

2.1. Overview of test loop

The high-pressure wet-gas test facility at NEL is based around an 200 mm nominal bore flow loop. A schematic diagram of the nominal facility arrangement for wet-gas tests is provided in Figure 1. Although nominally 200 mm diameter, the test section can accommodate line sizes ranging from 100 mm to 300 mm. The gas used for testing is oxygen-free nitrogen, supplied in 230 bar gauge cylinder banks. The facility typically operates at a nominal temperature of 20°C over a nominal pressure range of 10 to 63 bar gauge, which corresponds to a gas density range of 12.76 to 74.54 kg/m$^3$.

In Figure 1, the gas is driven around the flow loop by a 200 kW fully-encapsulated gas blower. In wet gas operation, the gas is drawn from the gas-liquid separator outlet by the blower and is then cooled using a chilled water supply flowing through shell and tube heat exchanger. The gas then passes through the reference gas flow meter and into the test line. The gas flow rate is controlled by varying the speed of the blower while the liquid flow rate is controlled by using two variable flow control valves located downstream of the liquid pumps. The maximum achievable dry gas volumetric flow rate is dependent on the size and type of reference or test flow meter installed.

![Flow test loop schematic](image)

**Figure 1.** Flow test loop schematic

In Figure 2, the Venturi tube is installed horizontally with the differential pressure, pressure and temperature transmitters mounted closely to the Venturi tube. All the transmitters are connected to NEL data acquisition and control system and the test is automatically run by the system.

2.2. Venturi details

Dimensions of the Venturi used in the experiment as indicated in Figure 3 below. The Venturi is sized according to ISO 5167:2003 [4]. The length of downstream pipe spool, which is located at the end of divergent cone, is 6D in order to get full pressure recovery profile at the downstream end of the Venturi tube. All the units in Figure 3 below are in mm.
The beta ratio, which is the ratio of internal throat diameter to the internal upstream pipe diameter, is 0.6. The nominal diameter of the Venturi is 250 mm and wall thickness of 12.75 mm, with pressure rating of ASME 900 lbs. The convergent angle of the inlet cone is 10.5° and divergent angle of the outlet cone is 7.5°. The two red dots on Figure 3 mark the positions where the pressure measurements are taken to obtain the differential pressure value. The yellow dots mark the positions where pressure measurements are taken to obtain the total permanent pressure loss across the Venturi tube. The green dots mark the positions where velocity measurement is taken for the volumetric flow rate calculation.

2.3. Test matrix
Table 1 gives the selected pressure and temperature test points, which are within the available range provided by the test facility. The selected test points are based on typical flow rates, pressure and temperature ranges for Venturi tubes used in the well test applications at offshore installation in Malaysia [5]. Although the result in this analysis will be used in natural gas application, nitrogen was selected for the test as it is considerably easier to handle and cost effective. In the earlier work by NEL, CFD did not pick up any appreciable effect from changing the nitrogen to other gases and the experiment did reveal the effect was small enough as to be largely ignored [6].
Table 1. Test matrix for the experiment

| Test Point | Pressure | Temp | Gas Flowrate | Nitrogen Density | Gas superficial velocity |
|------------|----------|------|--------------|------------------|-------------------------|
| (-)        | (bar gauge) | (°C) | (m³/h)       | (kg/m³)          | (m/s)                   |
| 1          | 60.37    | 20.04| 1213         | 70.921           | 7.00                    |
| 2          | 60.36    | 20.06| 875          | 70.904           | 5.05                    |
| 3          | 60.30    | 19.97| 520          | 70.860           | 3.00                    |

2.4. Discharge coefficient

Discharge coefficient is calculated using Equation (1):

\[
Cd = \frac{4m\sqrt{1-\beta^4}}{\pi ed^2\sqrt{200000Dp_1\rho_1}}
\]  

(1)

where \(m\) is the reference gas mass flow rate (kg/s), \(\beta\) is the diameter ratio (-), \(\varepsilon\) is the gas expansibility (-), \(d\) is the throat diameter (m), \(\rho_1\) is the upstream gas density (kg/m³) and \(Dp_1\) is the measure of upstream-to-throat differential pressure (bar) [7].

3. CFD setup

3.1. Venturi model

The Venturi model is drawn using ANSYS Design Modeller. The model consists of the internal section of the Venturi tube. For analysis, 2D symmetry has been used for the numerical simulation. Pressure based solver and SIMPLE pressure-velocity coupling have been used as these are applicable for wide range of flow regimes and applicable for this analysis. It also requires less memory (storage) and allows flexibility in the solution procedure.

3.2. Turbulence model

For the analysis, SKE turbulence model has been selected for the numerical simulation. The SKE model solves for two variables: turbulent kinetic energy, \(k\) and rate of dissipation of kinetic energy, \(\varepsilon\). The wall functions are used in this model. The SKE model is selected because of its good convergence rate and relatively low memory requirements [8].

3.3. Near wall meshing

As the differential pressure is measured from the pressure taps at the wall of the Venturi tube, near wall numerical simulation is important. Inflation has been used for the near wall meshing. Based on \(y^+\) equals to 1, the first layer thickness for the inflation setup is calculated for each of the test cases. The selected first layer thickness is 0.0011 mm as this is the smallest calculated first layer thickness from the test cases.

3.4. Convergence analysis

The numbers of elements ranges from 100,000 to 1,350,000 are used in the numerical analysis. The convergence for both cases has shown a decrease in residuals by two orders of magnitude for epsilon and continuity which indicates good qualitative convergence. The number of elements of 650,000 has been selected for the economics of computation as shown in Figure 4.
4. Results and discussion

From Table 2, the error between the experimental and numerical results is only 0.937%. This shows that the model selected in the numerical analysis has been verified to be suitable in the subsequent simulation in designing the Venturi tube for the gas applications.

| Venturi Under Test | Average Discharge Coefficient From experiment | Average Discharge Coefficient From CFD-post |
|--------------------|----------------------------------------------|---------------------------------------------|
| Venturi 1          | 0.99366                                      | 0.984347                                    |

4.1. Pressure contour charts

Pressure contours in Figure 5 are to plot pressure distribution across the Venturi tube. Flow direction for all the plots in Figure 5 is from left to right. The results clearly show the effect of mass flow rate of gas on the pressure gradient as obtained for various test cases. The pressure values at the upstream of the Venturi and at the centre point of the throat cylinder are recorded to determine the differential pressure across the Venturi tube, which are then used in the flow calculation.

4.2. Velocity contour charts

It is clear from Figure 6 that the velocity is at maximum at the throat of the Venturi. The velocity also decreases according to the decrease in flow rates. Flow direction for all plots in Figure 6 is from left to right. Velocity values recorded from the contours below are used to calculate the volumetric flow rates.
4.3. Discharge coefficient and other observations

The recorded differential pressure and velocity values are tabulated for the discharge coefficient calculation as shown in Table 3.

Table 3. Calculated discharge coefficient from CFD-post data

| Test | Dp1 (From CFD-post) | Press. (bar) | Temp. (°C) | Density (kg/m³) | Expansibility | Velocity (m/s) | Mass Flow Rate (kg/hr) | Discharge Coefficient, Cd |
|------|----------------------|-------------|------------|-----------------|--------------|---------------|-----------------------|---------------------------|
| 1    | 0.12286              | 60.37       | 20.04      | 70.92           | 0.9987       | 7.00          | 86,030                | 0.97551                   |
| 2    | 0.06220              | 60.36       | 20.06      | 70.90           | 0.9993       | 5.05          | 62,057                | 0.98850                   |
| 3    | 0.02191              | 60.30       | 19.97      | 70.85           | 0.9997       | 3.00          | 36,855                | 0.98902                   |
In Table 3, while the pressure, temperature, density and expansibility are almost constant, it is interesting to note that decrease in flow rates resulting in the increase of calculated Cd values. It may suggest that, with additional test points at much lower flow rates may bring the Cd value closer to the result obtained by experiment.

In Table 4, although mass flow rate is directly proportional to Cd, the increase in mass flow rate does not register any increase in Cd. This indicates that the Dp₁ has a significant effect on the Cd calculation.

### Table 4. Mass flow rate versus discharge coefficient (or Dp₁ versus discharge coefficient)

| Test | Dp₁ (bar) | (% change) | Mass Flow Rate (kg/hr) | (% change) | Discharge Coefficient, Cd |
|------|-----------|------------|------------------------|------------|--------------------------|
| 1    | 0.12286   | 98%        | 86,030                 | 39%        | 0.97551                  |
| 2    | 0.06220   | 184%       | 62,057                 | 68%        | 0.98850                  |
| 3    | 0.02191   |            | 36,855                 |            | 0.98902                  |

In Table 5, analysis on pressure gradient increment against the mass flow rate increment indicates that for every 1% increase in mass flow rate, it will register 2.5 times percentage increase in pressure gradient. The analysis also indicates that for every 1% increase in mass flow rate, it will register 1% increase in velocity gradient. This pattern is consistent between test 1 and test 2, and between test 2 and test 3. This indicates that pressure has more effect on the computation compared to velocity.

### Table 5. Pressure and velocity gradients

| Test | Pressure (bar gauge) | Mass Flowrate (kg/hr) | Pressure Gradient (bar/m) | Velocity Gradient (s⁻¹) |
|------|----------------------|-----------------------|---------------------------|-------------------------|
| 1    | 60.37                | 86,030                | -0.26                     | 56.51                   |
| 2    | 60.36                | 62,057                | -0.13                     | 40.77                   |
| 3    | 60.30                | 36,855                | -0.05                     | 24.23                   |

From Table 5, it shows that increase in pressure will exponentially increase the pressure gradient. This means for every bar increase in pressure, pressure gradient will exponentially become steeper and this in return will increase flow velocity at the Venturi throat. By understanding the correlation between pressure gradient and pressure, the convergent cone of the Venturi can be designed to operate within the velocity limit at Venturi throat.

In Table 6, the recorded PPL follows similar trend as compared to the Dp₁. The PPL obtained is approximately 11% of Dp₁ for each of the tests for 7.5° divergent angle. This follows the same observation recorded in the previous analysed data [9]. The knowledge on this correlation between Dp₁ and PPL can be used as validation check on the CFD model.

### Table 6. Permanent Pressure Loss (PPL)

| Test | Dp₁ (bar) | Mass Flow Rate (kg/hr) | PPL (bar) |
|------|-----------|------------------------|-----------|
| 1    | 0.12286   | 86,030                 | 0.01323   |
| 2    | 0.06220   | 62,057                 | 0.00699   |
| 3    | 0.02191   | 36,855                 | 0.00266   |

5. Conclusion

In the investigation, the flow through Venturi tube was analysed with SKE turbulence model. The results from the numerical simulation had been used to obtain the Cd values for each of the tests. The
Cd values obtained from simulation for test 1, test 2 and test 3 were 0.97551, 0.98850 and 0.98902, respectively, which averaged to 0.984347. Test 1 resulted in the lowest Cd value whereas Test 3 provided the maximum Cd value. The error between the experimental and numerical results of 0.937% indicates the SKE turbulence model agreed with the experimental data. The analysis also indicated that pressure has more computational impact compared to velocity, and by understanding the correlation between pressure gradient and pressure, the convergent cone of the Venturi can be designed to operate within the velocity limit at Venturi throat. From the analysis and results obtained, it is shown that the model selected in the numerical had been verified to be suitable in the subsequent simulation in designing the Venturi tube for the gas applications.

Acknowledgments
Authors acknowledge the assistance from FRGS grant 5524612, as well as support from KROHNE (M) Sdn. Bhd. and NEL.

References
[1] Jamieson A W, Johnson P A, Spearman E P and Sattary J A 1996 Proc. 14th North Sea Flow Measurement, Peebles, Scotland
[2] Van Weers T, Van Der Beek M P and Landheer I J 1998 Proc. 9th International Conference on Flow Measurement, FLOMEKO, Lund, Sweden 203-7
[3] Reader-Harris M J, Barton N, Brunton W C, Gibson J J, Hodges D, Nicholson I G and Johnson P 2000 Proc. 18th North Sea Flow Measurement, Perthshire, Scotland
[4] International Organization for Standardization 2003 Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full, Part 4: Venturi tubes BS EN ISO 5167-4:2003
[5] KROHNE (M) Sdn. Bhd. 2015 Venturi tubes installation reference list in Malaysia
[6] Reader-Harris M J, Gibson J J and Hodges D 2007 Proc. 6th South East Asia Hydrocarbon Flow Measurement, Kuala Lumpur, Malaysia
[7] Fang L and Zhang T 2008 Chinese Journal of Chemical Engineering 16 320-4
[8] Sowjanya V and Jie C 2010 Applied Mathematical Modelling 34 1458-66
[9] Reader-Harris M J 2015 Orifice plates and venturi tubes (Cham: Springer International Publishing)