IOP Conf. Series: Materials Science and Engineering **243** (2017) 012036 doi:10.1088/1757-899X/243/1/012036

# CFD simulation of flow through an orifice plate

**M M Tukiman[1], M N M Ghazali[1], A Sadikin[1], N F Nasir[1], N Nordin[1], A Sapit[1], M A Razali[1]**

[1]Department of Energy and Thermofluid Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn, Malaysia

E-mail: azmah@uthm.edu.my

**Abstract**. In this present paper, the commercial Computational Fluid Dynamics (CFD) is used to predict the flow features in the orifice flow meter. Outcomes of the CFD simulations in terms of profiles of velocity and pressure are discussed in detail. It is observed that the flow is jet-like flow in the core region and the presence of recirculation, reattachment and shear layer regions flow features downstream the orifice. The location of vena-contracta was also estimated from CFD simulations. These results are consistent with other published data.

## 1. Introduction

The orifice plate is very inexpensive for it is just a flat plate and a thin orifice plate. It is also very easy to install in the pipeline. It works on simple principle of using effects of velocity and pressure variation caused by reduction of the available area for flow. Orifices are widely used in flow metering because of their ruggedness, simple mechanical construction, and other known advantages [1]. The study on effects of orifice to pipe diameter ratio are extensive. The effects of ratio on the mass transfer rate behind the orifice numerically studied by [2] and [3]. The characteristic length and velocity scales in different regions of orifice flows using PIV was studied by [4] and found out that the position of the vena contracta shows a slight orifice to pipe diameter ratio dependence.

CFD tools are also widely used in modelling and analyzing orifice meters [5-8]. The flow through conventional single-hole orifice meter and integrated CFD simulation with measurements to analyze oscillating air flow through an orifice in a circular pipe was studied using used Open FOAM-1.6 by [1]. Some studies have focused on obtaining the associated discharge coefficients [9] and pressure drops [10]. Although the orifice plate is cheap, robust and easy to implement, but one disadvantage of this meter is it maximizes form friction. And it is very serious that by using this kind of meter a large percentage of pressure drop is unrecovered. The fluid velocity is increased at the opening of the orifice plate and not much energy is lost but as it flows through and starts slowing down, much of the excess energy is lost. With the above distinct advantages of a flow meter of high industrial importance, it is necessary to understand the flow pattern of orifice meter to further improve its performance in terms of flow measurement with better accuracy. Therefore, in this paper, CFD simulations are carried out to predict the flow pattern and vena contracta in the orifice meter. The outcomes of the CFD simulations in terms of velocity profile and pressure profile. CFD would candidate as a valuable engineering tool for the design and use of these devices in all those situations, quite common in the oil and gas fields, where experiments may be too complex or time consuming.

## 2. Governing equations and modelling assumptions

The governing equations (continuity and momentum) with the appropriate Reynolds stress closure need to be solved. In the present work, standard *k-epsilon* turbulence model by [11] has been used. The *k-epsilon* turbulence model is simple to use, cheap, most widely validated and has excellent performance for many industrial flows [12]. Since the problem is assumed to be steady and incompressible flow, time dependent parameters are dropped from the equations. Below shown the governing equation for the flow in orifice plate;

Conservation of mass;

$$\nabla.\left(\rho\overline{V}\right)=0 \tag{1}$$

*y*-momentum;

$$\nabla.\left(\rho v\overline{V}\right)=-\frac{\partial p}{\partial y}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z} \tag{2}$$

*z*-momentum;

$$\nabla.\left(\rho w\overline{V}\right)=-\frac{\partial p}{\partial z}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z} \tag{3}$$

## 3. Geometry, grid and computational domain

A set of CFD simulations is performed to investigate the flow through an orifice meter with a diameter throat of 5 mm and the length is 2 mm. The length of the upstream flow is 246 mm and total length of the pipe is 494 mm. The pipe has 12.30 mm inlet diameter. Adequate length at the upstream and downstream of the orifice plate is to avoid the effect of patched boundary conditions on the flow through the orifice. This also to provide fully developed turbulent flow in the upstream of the orifice plate. The model of the orifice is shown in figure 1.

. The meshing gave a total of 595 000, nodes and had 740,000 elements that consisted of unstructured hexahedral. It was ensured that the viscous sublayer ($y^+$ <5) and several nodes in the buffer and turbulent zones for fine meshes in the orifice region. The working fluid is water. Static pressure is assigned to the outlet pipe in order to obtain relative pressure drop between inlet and outlet. The inlet velocity profile is assumed to be uniform and set to subsonic flow of 0.38 m/s. The pipe is

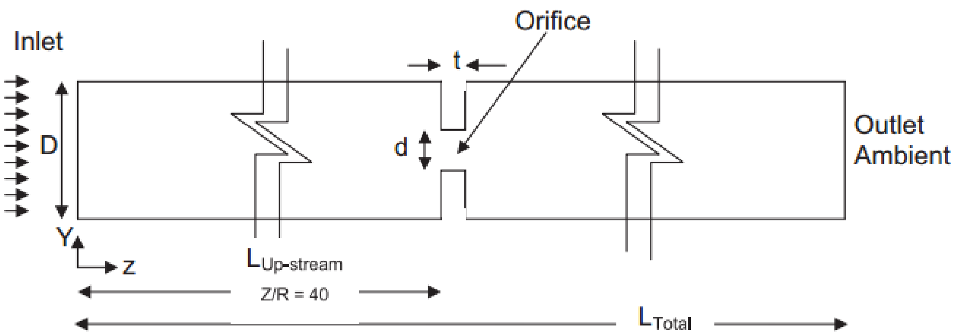



**Figure 1.** The orifice geometry (orifice is at z/R = 40 from inlet).

modelled as solid wall with no slip condition. The *k-epsilon* model was chosen for the turbulence model and the numerical accuracy was set to first order. All the discretized equations were solved in a segregated manner with the SIMPLE (Semi Implicit Pressure Linked Equation) algorithm. The simulation was run until the residual of the pressure and velocities were less than 0.00001.

## 4. Results of calculations

Velocity contour plot and velocity vectors are given in figure 2. As we can see form the figure, as the water flows from a narrowing orifice plate, the flow forms a free flowing jet in the downstream fluid. The velocity is at its maximum. The separation of boundary layer is seen at the downstream side of orifice plate. The turbulent and wake region, together with recirculation zones can be seen just downstream of orifice meter. The flow pattern for orifice flow is well captured in the numerical simulations. These observations are excellent agreement with [1].

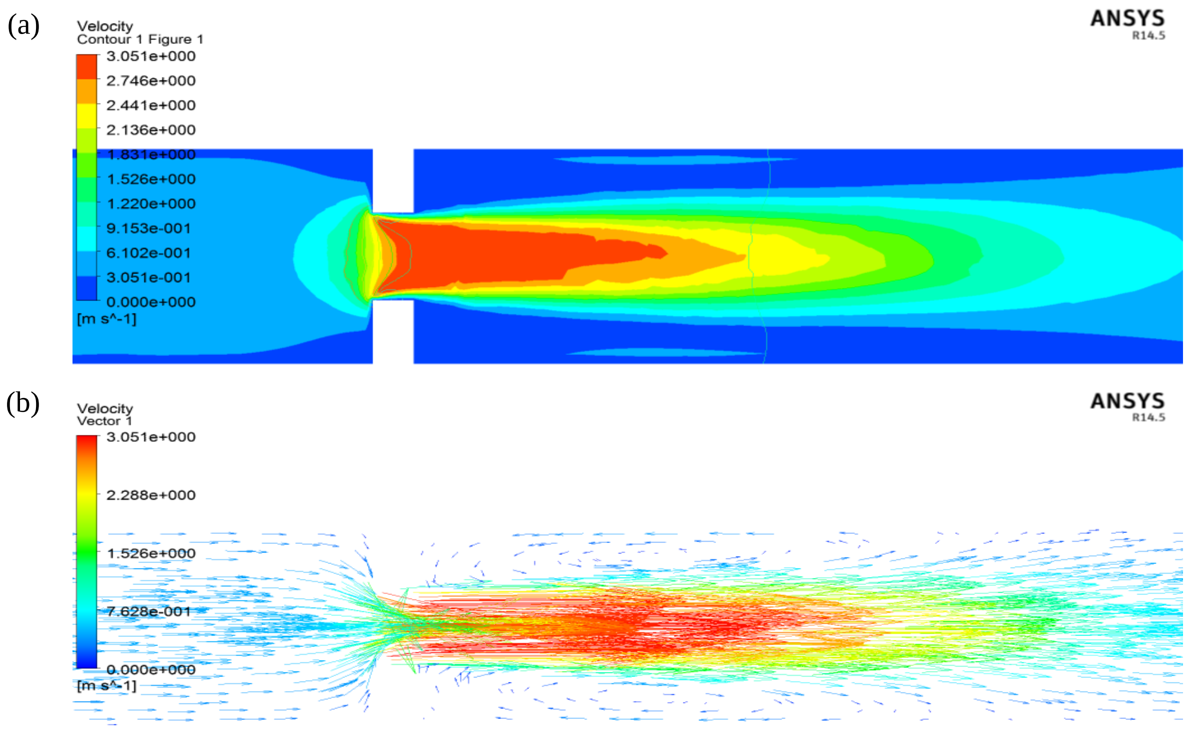



**Figure 2.** (a) Velocity contours and (b) velocity vectors

Figure 3 shows the velocity profile upstream of orifice at z = 0.15 m from the entrance. The strong velocity gradient near the wall because of the no-slip condition is observed. This region is often very thin, and it is then called a boundary layer. The boundary layers grow in thickness as more and more of the fluid is affected by viscous friction originating from the gradients set up by the no-slip condition. The layers meet in the middle and merge, and the flow reaches an asymptotic state called fully-developed flow. This figure also indicates that the fully turbulent flow is developed at the upstream of the orifice meter. This shows that the length is adequate for pipe upstream for fully developed flow. Nonetheless, the length for pipe downstream of the orifice meter is also adequate for the recovery of pressure.

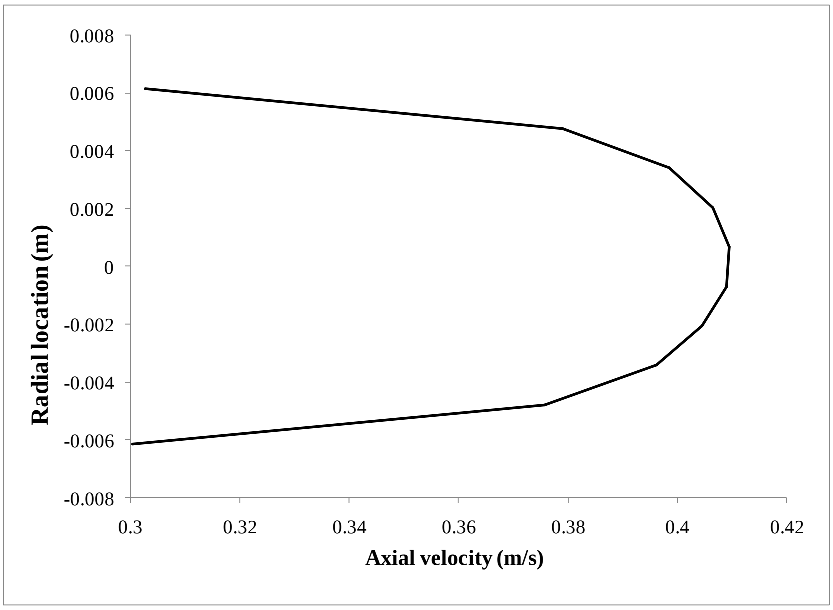
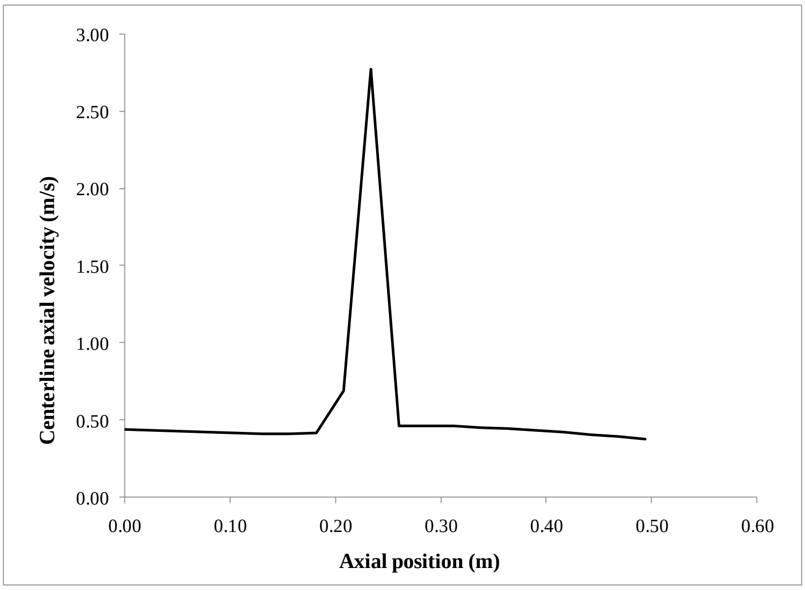



**Figure 3.** Radial profile velocity upstream of orifice at axial distance of *z* = 0.15 m from entrance

Centerline axial velocity profiles and centerline pressure profile are given in figure 4 (a) and (b) respectively. Centerline axial velocity increases as the flow approaches the throat of orifice meter. The further it travels downstream of orifice meter and reaches at its maxima at z = 0.234 m. This point of maximum velocity is called vena-contracta. Beyond this point the velocity decreases. As for pressure, as we can see from figure 4 (b), energy has to be conserved at all the points in the domain, accordingly the pressure decreases when the flow approaches orifice meter, reaching minimum at vena-contracta and starts recovering as the flow moves further downstream. This is supported by the work of [1].

(a)



(b)

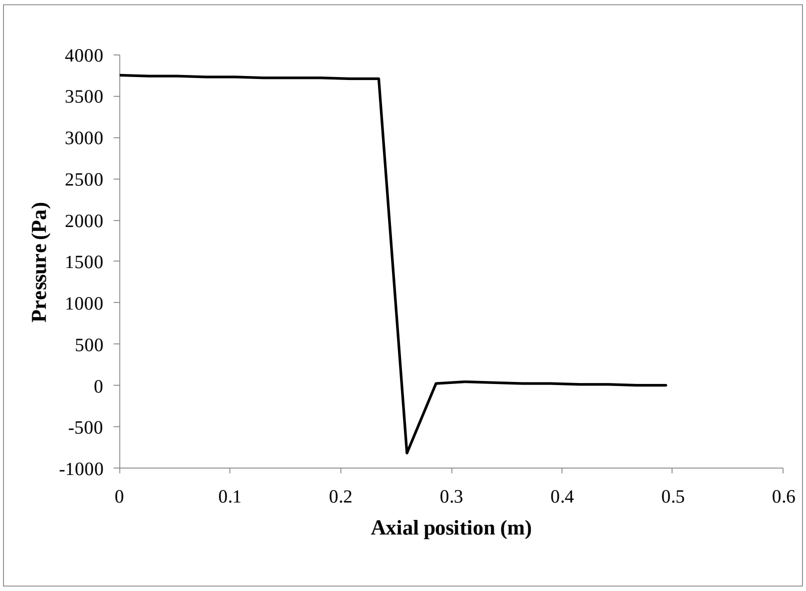



**Figure 4.** (a) Centerline axial velocity and (b) centerline pressure profile

## 5. Conclusion

The computational fluid dynamic simulations for orifice meter in a pipeline was carried out. The results agree well with the published data in terms of flow pattern, velocity profiles and pressure profile. The location of vena-contracta was estimated from CFD simulations. It is believed that flow physics, such as the location of the vena contracta and characteristic length and velocity scales in orifice flow, are very interesting for researchers and engineers in the study of flow metering, particularly the effects of the ratio on these flow physics. It is also concluded that the CFD technique can be used as an alternative and cost effective tool towards replacement of experiments required for estimating discharge coefficient, empirically. Further work to use orifice meter on different liquids and gases at different beta ratios are to be carried out in future.

## Acknowledgement

The author would like to thank both Universiti Tun Hussien Onn Malaysia (UTHM) and Ministry of Higher Education of Malaysia for the financial support through Research Accultration Grant Scheme (Vot R071).

## References

[1] Shah M S, Joshi J B, Kalsi A S, Prasad C S R and Shukla D S 2012 Analysis of flow through an orifice meter: CFD simulation Chem. Eng. Sci. 71 300-309

[2] El-Gammal M, Ahmed W H and Ching C Y 2012 Investigation of wall mass transfer characteristics downstream of an orifice Nucl. Eng. Des. 242 353-360

[3] Ahmed W H, Bello M M, El Nakla M and Al Sarkhi 2012 Flow and mass transfer downstream of an orifice under flow accelerated corrosion conditions Nucl. Eng. Des. 252 52-67

[4] Shan F, Liu Z, Liu W and Tsuji Y 2016 Effects of the orifice to pipe diameter ratio on orifice flows Chemical Engineering Science 152 497-506

[5] Reis L, Carvalho J, Nascimento M, Rodrigues L, Dias F and Sobrinho P 2014 Numerical modeling of flow through an industrial burner orifice Appl. Therm. Eng., 67 201-213

[6] Shaaban S 2014 Optimization of orifice meter's energy consumption Chem. Eng. Res. Des. 92 1005-1015

[7] Singh V and Tharakan T J 2015 Numerical simulations for multi-hole orifice flow meter Flow Meas. Instrum. 45 375-383
[8] Kumar P and Bing M W M 2011 A CFD study low pressure wet gas metering using slotted orifice meters, Flow Meas. Instrum. 22 (1) 33-42
[9] Reader-Harris M 2015 Orifice Discharge Coefficient, Orifice Plates and Venturi Tubes, Springer 127-186
[10] Cioncolini, Scenini F and Duff J 2015 Micro-orifice single-phase liquid flow: pressure drop measurements and prediction Exp. Therm. Fluid Sci. 65 33-40
[11] Launder and Spalding D 1974 The numerical computation of turbulent flows Comput. Methods Appl. Mech. Eng. 3 269-289
[12] Versteeg H K and Malalasekera W 1995 An introduction to computational fluid dynamics: The finite volume method (England:Pearson Education Limited)