International Journal of Engineering and Technical Research (IJETR)
ISSN: 2321-0869, Volume-3, Issue-5, May 2015

Prediction of Performance Characteristics of Orifice Plate Assembly for Non-Standard Conditions Using CFD

Karthik G S, Yogesh Kumar K J, V Seshadri

***Abstract*—Performance characteristics of orifice plates for variation in different geometrical parameters are analyzed with the help of computational fluid dynamics (CFD) for non-standard conditions with water as working fluid. In the present work, A CFD tool, ANSYS, FLUENT has been used to predict the variation of Coefficient of Discharge (C_d) and the analysis for standard sharp edge orifice meter has been done for different plate thicknesses (3mm, 5mm, 10mm and 15mm) in a pipe of 50mm diameter and the effect of Pipe diameter on Coefficient of Discharge (C_d) has been studied in detail. The value of Coefficient of Discharge depends on the type of flow, pressure tappings, contour of the obstruction and it is a function of Reynolds number. The results and outcomes of the study are presented with velocity and pressure contours. The effect of non-standard conditions on Coefficient of Discharge (Cd) has been discussed in detail.**

***Index Terms*— Coefficient of Discharge (Cd), CFD, Reynolds number, ANSYS, FLUENT, Orifice Meter**

I. INTRODUCTION

Flow measurement is essential part in many industries today. The flow rate is to be found out at a check point which can be achieved by the device called a flow measuring device or a flow meter. The most common type used is the Differential Pressure type flow meters, sometimes also called the Head-Loss or Obstruction type flow meters. In any Obstruction type flow meter, the flow rate is calculated by measuring the pressure drop over an obstruction inserted in the path of the flow.

An orifice plate is a thin plate with a hole in it which is placed inside the pipeline for the determination of flow rate of the flowing fluid. The orifice plate may also be used for reducing pressure or for restricting a flow in case of restriction orifice plate. When a fluid passes through the orifice; its pressure builds up slightly upstream of the pipe and the fluid is forced to pass through the hole. The velocity increases and the fluid pressure decreases as the area reduces due to the presence of obstruction. A little downstream of the orifice the flow reaches its point of maximum convergence called the vena Contracta where the velocity reaches its maximum value. This is where the pressures are noted down by the tappings.

We know that not always standard dimensions and procedures can be used for an application. Industries have processes which demand or require non-standard operating range. There may be aging factors due to years of usage and erosion, operating under high pressures, non-accurate manufacturing or requirement of non-standard operating range for the industrial applications.

Karthik G S, M.Tech Student, Thermal Power Engineering, MIT-Mysore
Yogesh Kumar K J, Assistant Professor, Department of Mechanical Engineering, MIT-Mysore
V Seshadri, Professor (Emeritus), Department of Mechanical Engineering, MIT-Mysore

To scientifically analyze them is a challenge. This attempt is to see if CFD tool can be used for accurately calculating Cd in such cases and to develop a methodology that could be used for solving CFD problems under non-standard conditions. The results are positive if the CFD tool is properly handled.

Extensive work has been done in previous studies and many have analyzed the effects of various parameters on the Coefficient of Discharge. Singh et.al [1]conducted a detailed study on the effects of small diameter orifice meters, with different beta ratios and plate thickness on Cd using GAMBIT and ANSYS Rahman et.al [4]have done an experimental work in hydraulic lab for different beta ratios using a setup and they analyzed and gave a relationship of beta ratio with Re and Cd. Naveenji Arun et.al [6] have studied the flow of non-Newtonian fluids using CFD and plotted the variation of Cd with different concentrations of fluids and obtained that C_d increases exponentially with increase in Reynolds Number.

II. WORKING PRINCIPLE OF ORIFICE METER ASSEMBLY:

The working principle of any obstruction type flow meter is based on Bernoulli's Equation given by

$$\frac{p_1}{\rho g} + z_1 + \frac{{v_1}^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{{v_2}^2}{2g}$$

The volumetric flow rate can be expressed by the equation:

$$Q_{Theoretical} = v_2 A_2 = \frac{A_2}{\sqrt{1-\beta^4}} \sqrt{\frac{2g \times (p_1 - p_2)}{\rho g}}$$

Where d1and d2 are the diameters of the pipeline and the orifice opening, p1and p2 are upstream and downstream pressures, then the flow rate can be obtained using above equation by measuring the pressure difference (p1- p2).

The flow expression obtained in the above equation is not accurate in a practical case since it does not take into account the real fluid effects and hence an empirical factor called discharge co-efficient (Cd) is incorporated. Thus the equation becomes

$$Q_{Actual} = v_2 A_2 = \frac{C_d A_2}{\sqrt{1-\beta^4}} \sqrt{\frac{2g \times (p_1 - p_2)}{\rho g}}$$

III. DISCHARGE COEFFICIENT OF ORIFICE PLATE:

Cd is defined as the ratio of the actual flow to the theoretical flow and is always less than one. There are many reasons due to which the actual flow rate is less than the ideal one. Mainly the assumptions of frictionless flow and incompressible flows are not always valid. The extent of friction depends on the Reynolds number (Re). As a result, the correction factor Cd<1, has to be incorporated.

Calculation of Cd can be done using the formula specified in ISO-5167. The discharge coefficient, Cd is given by the Reader-Harris/Gallagher Equation which is purely empirical and is based on extensive experimental data. Their correlation is given by,

$$\begin{aligned} c_d &= 0.5961 + 0.0261\beta^2 - 0.216\beta^8 \\ &+ 0.000521[10^6\beta/Re_D] \\ &+ [0.0188 + 0.0063A]\beta^{3.5}[10^5/Re_D] \\ &+ [0.043 + 0.080e^{-10L_1} \\ &- 0.123e^{-7L_1}][1 - 0.11A]\left[\frac{\beta^4}{1-\beta^4}\right] \\ &- 0.031[M_2 - 0.8M^{1.1}{}_2]\beta^{1.3} \end{aligned}$$

If D<71.12mm (2.8 in), the following term should be added to the equation.

$$+0.011(0.75 - \beta)\left[2.8 - \frac{D}{25.4}\right]$$

Where D is the diameter of pipe in mm

Diameter ratio, β =d/D

Re_D =Pipe Reynolds number

A= [19000β/Re_D] 0.8

M2=2L2/ (1-β)

L1 and L2are lengths from the faces to tappings and depend on the type of taps used.

The above equation has its own limitations. The equation is completely empirical and is evaluated only by experimental results.

It is only applicable when pipe diameter is within 50mm<D<1000mm when diameter ratio is within 0.2< β<0.75, plate thickness between 0.005D and 0.02D, bevel angle of $45^0 \pm 15^0$. Further the upstream edge of the orifice hole should be sharp and a radius of curvature not greater than 0.0004d.

The geometry and dimensions should be as per ISO 5167 standards for the equation to be applicable. For any other non-standard cases, the equation is invalid.

IV. CFD & Mathematical Model

CFD is the most popular tool that is used nowadays to analyze a problem. CFD gains its popularity due to its capability to produce good results without conducting any experiments.

The governing equations used by the CFD code Fluent that has been used in the present study are

- The continuity equation or the conservation of Mass Equation for two dimensional incompressible flows.

$$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

- The Navier Stokes Equation or the conservation of Momentum Equation for two dimensional incompressible flows.

$$\rho\frac{du}{dt} = \rho g - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

$$\rho\frac{dv}{dt} = \rho g - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

In the case of turbulent flow, additional equations need to be solved depending on the turbulence model that is used in the computational analysis. The details of these models are not explained in detail since they are available in the standard literature.

The standard k-ε model (Launder and Spalding, 1974) is used in the simulations carried out and the model has two model equations, one for k (turbulent kinetic energy) and one for ε (rate of viscous dissipation).

The standard k-ε model uses the following governing equations for k and ε shown below.

$$\frac{\partial(\rho k)}{\partial t} + \mathrm{div}(\rho k U) = \mathrm{div}\left[\frac{\mu_t}{\sigma_k}\mathrm{grad}\ k\right] + 2\mu_t S_{ij} S_{ij} - \rho\varepsilon$$

$$\begin{aligned} &\frac{\partial(\rho\varepsilon)}{\partial t} + \mathrm{div}(\rho\varepsilon\ U) \\ &= \mathrm{div}\left[\frac{\mu_t}{\sigma_\varepsilon}\mathrm{grad}\ \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t S_{ij} S_{ij} \\ &- C_{2\varepsilon}\,\rho\frac{\varepsilon^2}{k} \end{aligned}$$

Where the eddy viscosity, $\mu_t$is given by the following equation

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where, C_μ is a dimensionless constant.

The k-ε model is the simplest turbulence model which only requires initial and boundary conditions well established and most widely validated turbulence model. It provides excellent results for many industry oriented flows. But the model produces poor results in case of some unconfined flows with large strains, swirling flows, recirculation and rotating flows etc. thus the k-ω model (Wilcox, 1988, 1993a,b, 1994)can be used as the best alternative which uses turbulence frequency ω as second variable.

The flow analysis is done using the help of commercially available CFD tool, ANSYS, FLUENT which uses finite volume approach to discretize the problem.

V. Validation

Validation is an important step in any problem solving technique. In CFD, validation needs to be done to prove that the mesh density used, type of mesh used, boundary conditions, turbulence model used in the simulation are proper and the results obtained are accurate. The validation may be done by comparing the results obtained by CFD with the experimental results obtained by conducting experiments or by comparing it with the available standard values in the ISO-5167 and BS-1042 standards.

The validation for the present problem was done for the flow through standard orifice plate of pipe diameter 50mm and Reynolds number of 1, 00,000 for beta ratio of 0.5 with inlet Velocity of 2m/s and exit gauge pressure of 0Pa.

The geometry of the above problem is as shown in Fig 1.

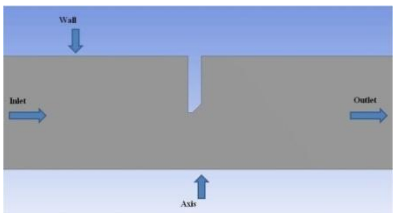
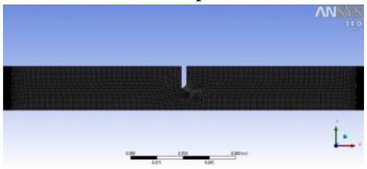


Fig 1: Geometry and Boundary conditions of Standard Orifice plate.



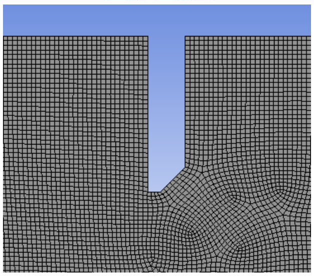


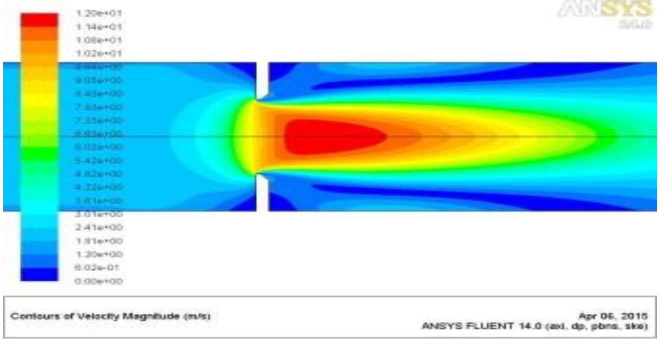
Fig 2: Mesh details of Standard Orifice Plate.

The modeling was done in the ANSYS Workbench14.0. The model was divided into 3 parts for efficient meshing. The upstream length was chosen to be 15D and the downstream length of 20D was incorporated to avoid the interference of the boundary conditions. The Grid independence test was done starting from 50000 elements to 300000 elements to ensure the minimum mesh density required to meet the required accuracy of Cd. It was observed that after 140000 the value of Cd remained almost constant and hence the meshing was done using Quadrilateral elements of 1, 40,000elements as shown in Fig 2. Fine meshing was done near the orifice plate due to the computational limits and fine wall meshing near the wall to capture the variations. The necessary boundary conditions at the inlet, outlet and wall were applied (Inlet- Velocity inlet, Wall- Wall/No slip condition, Axis-Axis, Outlet –Pressure outlet). Problem was considered to be axis symmetric. Second order upwind discretization and SIMPLE scheme was utilized for solving. Residuals or convergence criteria of 10^{-6} was employed with double precision solver. Standard k-ε model was used in the computations.

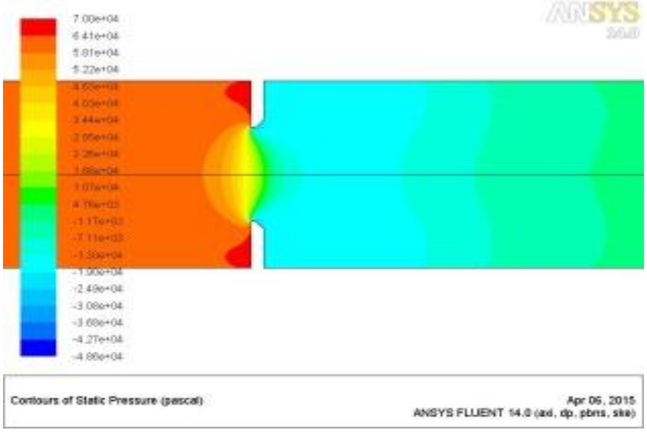
The coefficient of discharge (further referred to as C_d) for flange and corner taps were calculated using the pressure readings obtained at the respective pressure taps in CFD. The results obtained in CFD were validated against the ISO -5167 standard values and are shown in Table -1 and the respective velocity and pressure contours and plots are shown in Fig 3.

Turbulent Flow, Re=100,000		
C_d	Flange taps	Corner Taps
Value of C_d from ISO-5167 Standards	0.6082	0.6069
Value of C_d obtained by CFD	0.624	0.624

Table 1: Validation results for Standard Sharp Edge Orifice Plate.



Velocity Contour: Max Value :1.20e+01 m/s; Min Value: 0 m/s



Pressure Contour: Max Value : 7.00e+04 Pa, Min Value: -4.86e+04 Pa

Fig 3: Velocity and Pressure Contours Flow through Standard Orifice Plate.

The above results show that the computed values of C_d are somewhat higher than the values given in standards. Further calculations using different turbulence models have shown that the choice of model is also very important. The computations made with K-ω SST model gave the values of C_d as 0.6051 for flange taps and 0.6054 for corner taps. Thus the computed and standard values agree with each other within the uncertainty limits of 0.6% mentioned in the ISO-5167 standards. Thus the computational methodology can be considered as validated.

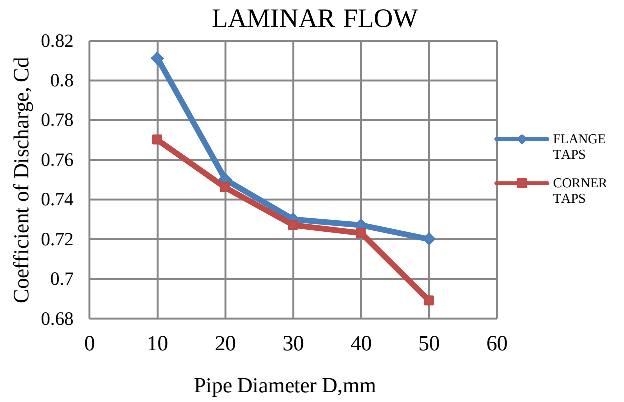
VI. Results and Discussions

EFFECT OF PIPE DIAMETER ON C_D

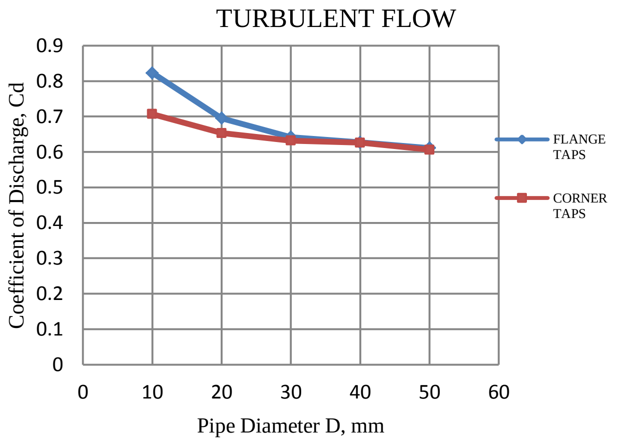
As per ISO 5167, the minimum pipe diameter for which the correlation for C_d is applicable is specified as 50mm.The effect of pipe diameter on C_d when it becomes less than 50mm has been studied in this section. The ranges of diameters selected for the study are from 50mm to 10mm and the diameter ratio is 0.5. Computations have been done for two Reynolds numbers namely 200 (Laminar Regime) and 100000 (Turbulent Regime).The obtained CFD results for various pipe diameters are tabulated in Table-2, and the corresponding velocity and pressure contours for laminar and turbulent cases are shown in Fig 4.As the diameter of the pipe is decreased, the conclusion that one can draw is that C_d increases gradually. The variation in flange taps is more as compared to the variation observed in Corner taps. In flange taps, the pressure tappings are located at a distance of 1 inch (25.4mm) and in corner taps the pressure taps are located at the plate corners and are independent of pipe diameter. Thus, in the case of flange taps the downstream pressure tap in relation to pipe diameter moves farther away from the orifice plate as the pipe diameter is decreased. Hence the pressure recovery will increase with decrease in diameter leading to a lower value of pressure differential. Consequently the value of C_d would correspondingly increase. This has been supported by the computations. However this effect is subdued in the case of corner taps. Hence it can be concluded that the equation given in the standards to calculate C_d is not applicable to the diameters below 50mm.The extent of error will keep on increasing with decreasing diameter.

Serial No.	Diameter of the pipe , D, mm	C_d for Laminar Flow, Re =200		C_d for Turbulent Flow, Re=100,000	
		Flange Taps	Corner Taps	Flange Taps	Corner Taps
1	50	0.720	0.689	0.624	0.624
2	40	0.727	0.723	0.627	0.626
3	30	0.730	0.727	0.642	0.632
4	20	0.750	0.746	0.695	0.653
5	10	0.811	0.770	0.823	0.707

Table 2: Effect of Pipe Diameter on Coefficient of Discharge.

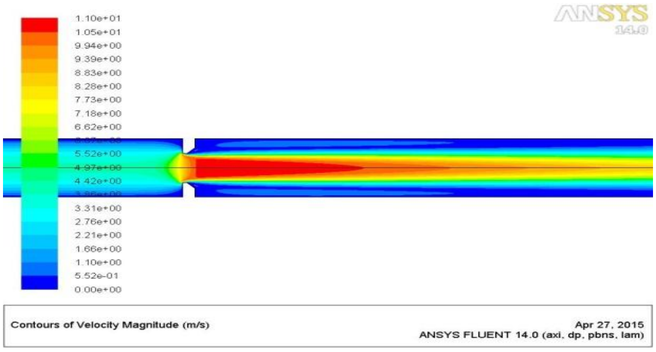


Variation of C_d for Laminar Flow

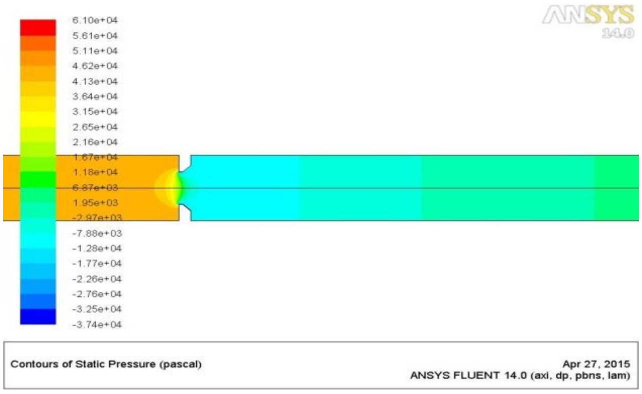


Variation of C_d for Turbulent Flow

Fig 4: Variations of Cd for Laminar and Turbulent cases for different pipe Diameters of Standard Orifice Plate.

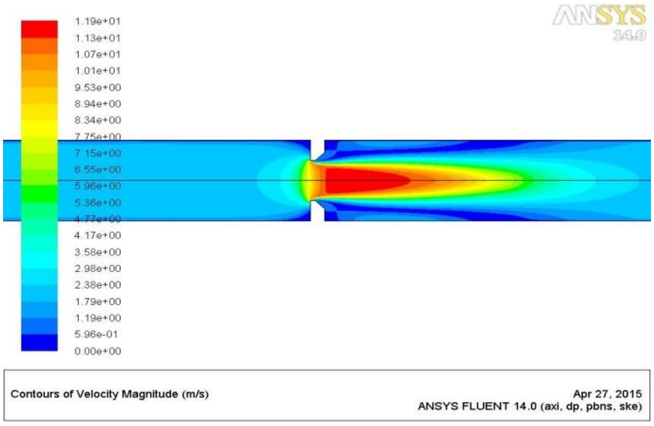


Velocity Contour: Max Value : 1.10e+01m/s ; Min Value: 0m/s

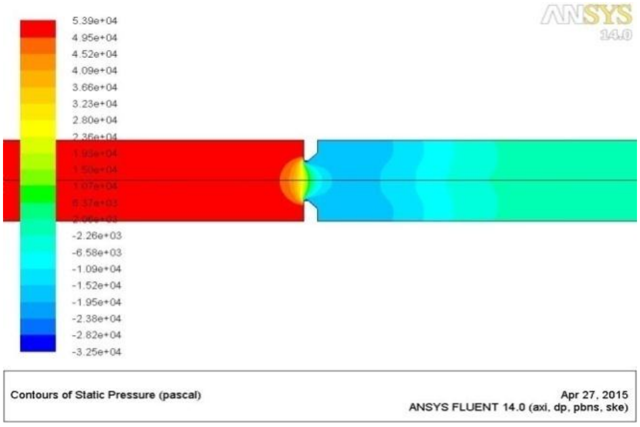


Pressure Contour: Max Value : 6.10e+04 Pa, Min Value: -3.74e+04 Pa

Fig 4: Contour plots of Non-Standard Orifice Plate (D=10mm) for Laminar flow.

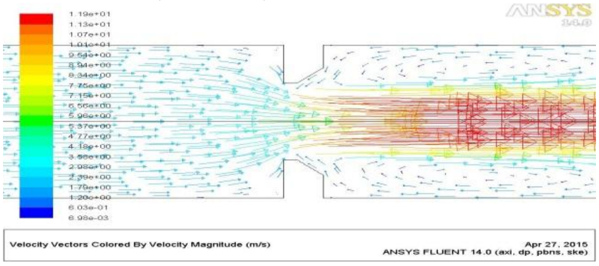


Velocity Contour: Max Value : 1.19e+01m/s ; Min Value: 0m/s



Pressure Contour: Max Value : 5.39e+04 Pa, Min Value: -3.25e+04 Pa

Fig 5: Contour plots of Non-Standard Orifice Plate (D=10mm) for Turbulent Flow.



Velocity Vector: Max Value : 1.19e+01 m/s; Min Value: 6.98e-03 m/s

Fig 6: Vectors plots of Non-Standard Orifice Plate (D=10mm) for Turbulent Flow.

EFFECT OF ORIFICE PLATE THICKNESS ON C_D

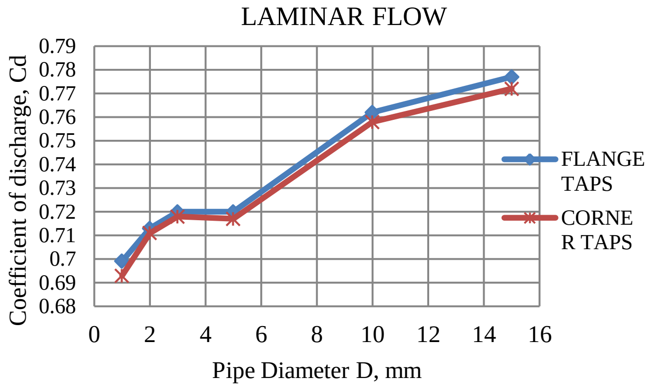
The maximum allowable thickness of the standard orifice plate as per standards for a 50mm diameter pipe is 3mm. In certain industrial applications, the pipeline pressure would be very high and to use an orifice plate to measure the flow rate in these cases higher plate thicknesses would be required in order to ensure structural safety. Thus for design considerations, the thickness of the orifice plates are increased. The effect of increased plate thickness on Cdis studied using CFD tool. The results given below are simulated for a 50mm diameter pipe and Beta ratio of 0.5 for both laminar and turbulent cases. Computations have been made for six plate thicknesses in the range 1mm to 15mm. The results are tabulated in table 3 for the two types of pressure tappings and at two Reynolds numbers.

The corresponding velocity and pressure contours for laminar and turbulent cases for a plate thickness of 5mm are given in Fig 8.

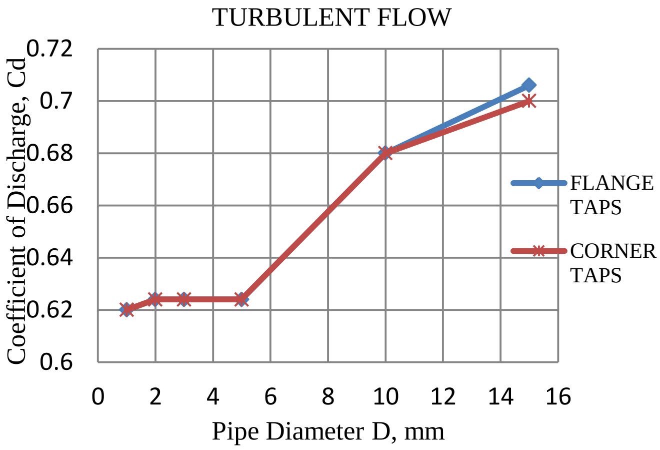
It is observed from the tabulated values that the computed values of C_d are independent of plate thickness as long as it is less than 5mm. In the case of thicker plates, the values of discharge coefficient increases with increasing thickness and the deviations are fairly significant. This shows that the correlation given in ISO 5167 could be used for plate thicknesses up to 5mm in a 50mm pipeline.

Serial No.	Thickness of the orifice plate, t, mm	C_d for Laminar Flow, Re =200		C_d for Turbulent Flow, Re=100,000	
		Flange Taps	Corner Taps	Flange Taps	Corner Taps
1	1	0.6991	0.693	0.620	0.620
2	2	0.713	0.711	0.624	0.624
3	3	0.720	0.689	0.624	0.624
4	5	0.72	0.717	0.624	0.624
5	10	0.762	0.758	0.68	0.68
6	15	0.777	0.772	0.706	0.700

Table3: Effect of plate thickness on Coefficient of discharge.

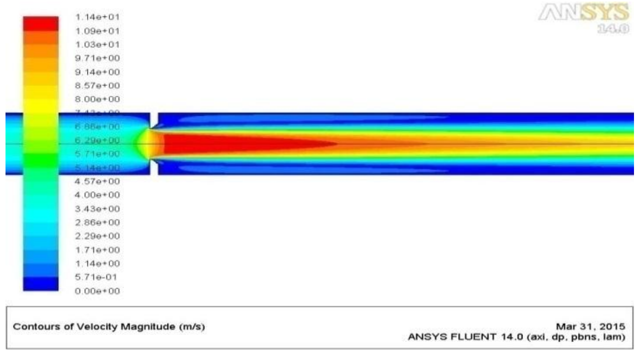


Variation of Cd for Laminar Flow

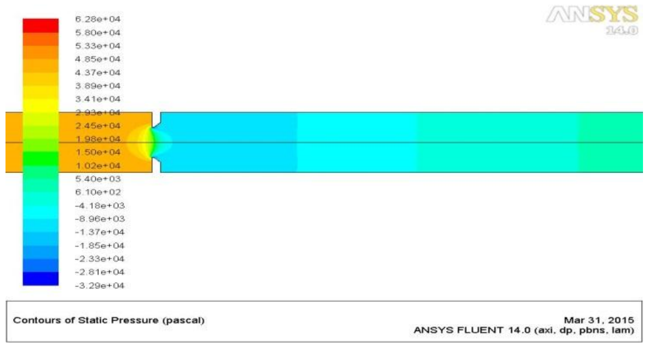


Variation of C_d for Turbulent Flow

Fig 7: Variations of Cd for Laminar and Turbulent cases for different Orifice Plate Thickness of Standard Orifice Plate.

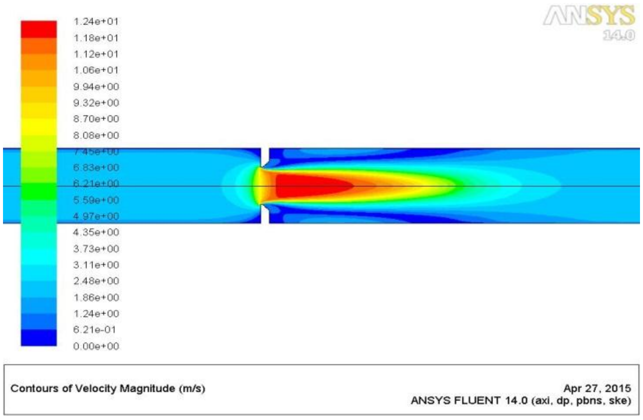


Velocity Contour: Max Value : 1.14 e+01m/s ; Min Value: 0m/s

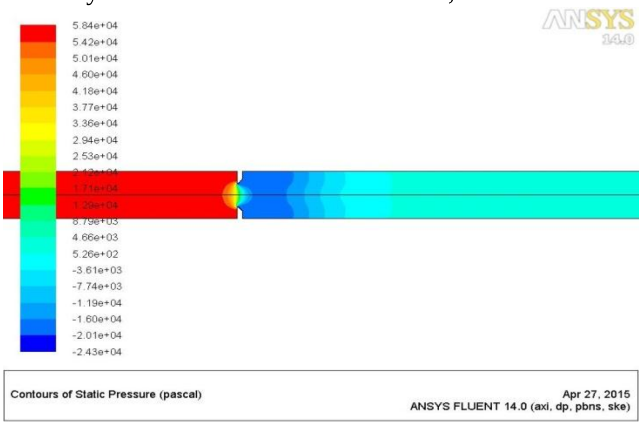


Pressure Contour: Max Value : 6.2/e+04 Pa, Min Value: -3.29e+04 Pa

Fig 8: Contour plots of Non-Standard Orifice Plate (t=5mm) for Laminar flow.

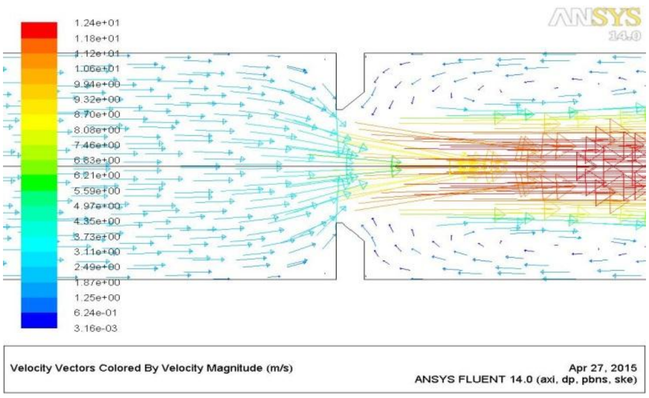


Velocity Contour: Max Value : 1.24e+01; Min Value: 0m/s



Pressure Contour: Max Value : 5.84e+04 Pa, Min Value: -2.43e+04 Pa

Fig9: Contour plots of Non-Standard Orifice Plate (t=5mm) for Turbulent flow.



Velocity Vector:Max Value : 1.24e+01 m/s; Min Value: 3.16e-03 m/s

Fig 10: Vectors plots of Non-Standard Orifice Plate (t=5mm) for Turbulent Flow.

VII. Concluding Remarks:

A CFD Methodology for analyzing flow through an orifice plate assembly has been presented. This methodology has been validated by comparing the results with the correlations available in the standards for a standard concentric square edged orifice plate. As long as turbulence model is carefully chosen, accurate estimation of C_d can be obtained from CFD analysis.

Further it has been demonstrated that CFD can also be used to predict the values of C_d even under non-standard conditions. It has been shown that if the pipe diameter is decreased below 50mm the value of C_d increases. Similarly, if the orifice plate thickness is larger than 5mm (in a pipe of 50mm diameter) the value of Cd can be substantially higher. Thus the study has demonstrated the utility of CFD in predicting the performance characteristics of the orifice plate assemblies.

References

[1] NaveenjiArun, Malavarayan S, Kaushik M "CFD Analysis on Discharge Coefficient during Non-Newtonian flows through Orifice Meter". International journal of Engineering Science and Technology. Vol 2(7), 2010.

[2] Harikrishna V Gohil, Mohammad AzimAijaz. "Finite Element Analysis of Pressure Drop in Orifice Meter". International Journal of Engineering Research and Technology. Vol 3, issue 3, march 2014.

[3] R K Singh, S N Singh, and V Seshadri."Performance Evaluation Of Orifice Plate Assemblies Under Non-Standard Conditions Using CFD" Indian Journal Of Engineering And Materials Sciences.vol. 17 December 2010.

[4] C B Prajapati, V K Patel, S N Singh, V Seshadri. "CFD Analysis of Permanent Pressure Loss for Different Types of Flow Meters in Industrial Applications" International Conference on Fluid mechanics and Fluid Power. 2010

[5] M N Rahman, R Biswas and W I Mafuz. "Effect of Beta Ratio on Reynolds Number on Coefficient of Discharge of Orifice Meter". Journal of agricultural and rural development. 2009..

[6] Manish S Shah, Jyeshtharaj B Joshi, Avtar S Kalsi, C S R Prasad, Daya S Shukla " Analysis of flow through an orifice meter : CFD Simulation" Chemical Engineering Science, 71 (2012).

[7] Indian Standard IS-14615 (part 1):1999 ISO-5167-1: 1991 Measurement of Fluid Flow by Means of Pressure Differential Devices.

[8] British Standard BS-1042 (Part 1: Pressure Differential Devices) Measurement of Fluid Flow in Closed Conduits.

[9] H K Versteeg and W Malalasekera "An introduction to Computational Fluid Dynamics, The Finite Volume Method, Second Edition"