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

Predictionof discharge coefficient of Venturimeter at low Reynolds numbers by analytical and CFD Method

Arun R, Yogesh Kumar K J, V Seshadri

Abstract- **The venturimeter is a typical obstruction type flow meter, widely used in industries for flow measurements. The ISO standard (ISO-5167-1) provides the value of discharge coefficients for the classical venturimeters in turbulent flows with Reynolds number above $2x10^5$. But in case of viscous fluids, venturimeters are sometimes operated in laminar flow rather than turbulent flow at Reynolds number below the range covered by the standards. In the present work, an attempt was made to study, prepare a computational model of a venturimeter, which can be used as an efficient and easy means for predicting the discharge coefficients at low Reynolds number. The computational fluid dynamics (CFD) software ANSYS FLUENT-14 has been used as a tool to perform the modeling and simulation of venturimeter. Simulation was carried out for a standard venturimeter and the results were validated with the standards. Further the focus of the study was directed towards flows with low Reynolds numbers commonly associated with pipe line transportation of viscous fluids. An analytical correlation for discharge coefficient in the laminar region was derived by considering the viscous losses that occur between two pressure taps. The results of the simulations show that the discharge coefficient decreases rapidly as the Reynolds number decreases. The results were compared with the analytically proposed equation to calculate Cd at low Re and also with the experimental data of Gordon Stobie [1]. The results obtained from all three modes of calculations were with an uncertainty of 0.9 per cent.**

Index Terms- **Venturimeter, Computational Fluid Dynamics (CFD), Coefficient of discharge (C_d), Low Reynolds number.**

I. INTRODUCTION

The flow meters are being widely used in the industries to measure the volumetric flow rate of the fluids. These flow meters are usually differential pressure type, which measures the flow rate by introducing a constriction in the flow. The pressure difference caused by the constriction is used to calculate the flow rate by using Bernoulli's theorem.

If any constriction is placed in a pipe carrying a fluid, there will be an increase in the velocity and hence the kinetic energy increases at the point of constriction. From the energy balance equation given by Bernoulli's theorem, there must be a corresponding reduction in the static pressure. Thus by knowing the pressure reduction, the density of the fluid, the area available for flow at the constriction and the discharge coefficient, the rate of discharge from the constriction can be calculated. The discharge coefficient 'C_d' is the ratio of actual flow to the theoretical flow. The widely used flow meters in the industries are orifice meter, venturimeter and Pitot tube. Venturimeter and orifice meter are more convenient and frequently used for measuring flow in an enclosed ducts or channels.

Venturi meters are commonly used in single and multiphase flows. The conditions encountered in metering viscous fluids can be beyond the range of applicability of the industry standards (ISO 5167-1). The ISO standard is limited to turbulent flows for Reynolds numbers (based on the upstream pipe diameter) above $2x10^5$. Metering viscous oils may involve laminar, rather than turbulent flow at Reynolds numbers below the range covered by the ISO standard. Little data are available on Venturi discharge coefficients in laminar flow. Hence CFD modeling was conducted to simulate flow through venturimeter at very low Reynolds number. The objectives was to gain additional insight into low Reynolds number flows in Venturi meters to evaluate how well CFD modeling and analytical calculation can predict the experimental results.

A. *Principle of Venturimeter:*

The venturimeter is an obstruction type flow meter named in honor of Giovanni Venturi (1746-11822), an Italian physicist who first tested conical expansion and contraction.

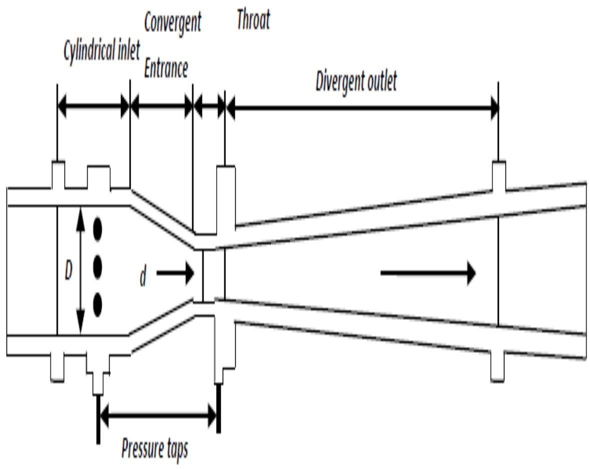


Fig.1: Venturimeter

The classical venturimeter consists of a converging section, throat and a diverging section as shown in the figure 1. The function of the converging section is to increase the velocity of the fluid and temporarily reduce its static pressure. Thus

Arun R, M.Tech Student, Thermal Power Engineering, MIT-Mysore.

Yogesh Kumar K J, Assistant Professor, Department of Mechanical Engineering, MIT-Mysore.

V Seshadri, Professor, Departmental of Applied Mechanics, IIT-Delhi, New Delhi.

the pressure difference between the inlet and the throat is developed. This pressure difference is correlated to the rate of flow of fluid by using Bernoulli's equation. As the theorem states that "In a steady, ideal flow of an incompressible fluid, the total energy at any point of fluid is constant, the total energy consists of pressure energy, kinetic energy and potential energy".

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 \qquad (1)$$

Because of the smoothness of the contraction and expansion section of venturi, the irreversible pressure loss is low. However, in order to obtain a significant measurable pressure drop, the downstream pressure tap is located at the throat of the venturimeter. In comparison of venturimeter with the orifice meter, the pressure recovery is much better for venturimeter than for the orifice plate, but there is no complete pressure recovery. Pressure recovery is measured as the pressure difference between inlet and outlet.

As per ISO 5167-1 [3], the mass flow rate in a Venturimeter (Q_m) is given by:

$$Q_m = \frac{C_d}{\sqrt{1-\beta^4}}\varepsilon\frac{\pi d^2}{4}\sqrt{2(p_1-p_2)/\rho_1} \qquad (2)$$

Where:
Cd = Venturimeter discharge coefficient
β = Venturimeter beta ratio, d/D
ε = Expansion factor (ε = 1, for incompressible flow)
d = Venturimeter throat diameter
D = Upstream pipe dia. of Venturi convergent section
P_1 = Static pressure at the upstream pressure tap
P_2 = Static pressure at the Venturi throat tap
ρ_1 = Fluid density at the upstream tap location

Equation "(1)" is based on the assumptions that the flow steady, incompressible, and inviscid flow (no frictional pressure losses) and uniform velocity profiles occur at the pressure tap locations. However in order to take into account the real fluid effects like viscosity and compressibility to empirical coefficients C_d and ε are introduced in the equation. In this paper we are considering only incompressible flow therefore ε = 1.

Over the years the venturimeters have been used for metering the different flows (liquid, gas, mixed flow). The performances of these meters in terms of value of discharge coefficient and pressure loss have been investigated by several researchers. Gordon Stobie et al [1] made a performance study on effect oferosion in a Venturi Meter with Laminar and Turbulent Flow and measurement of discharge coefficient at low Reynolds Number.Naveenji Arun et al [6] conducted a CFD analysis to predict the discharge co-efficient of venturimeter as a function of Reynolds number with different beta ratios for single phase non-Newtonian flows. C. L. Hollingshead et al [2], conducted experimental studies on discharge coefficients of venturi and validated using numerical analysis. T. Nithin et al [4] investigated the effect of divergence angle on the total and differential pressure drops for various values of angle of divergence and inlet velocities for the different venturi profiles.

In this present work, CFD modeling was conducted to simulate flow through venturimeter at very low Reynolds number and the objectives was to gain an additional insight into low Reynolds number flows in Venturi meters by using CFD and to propose an analytical correlation to predict the value of C_d at low Reynolds that match up with the experimental data.

II. CFD Modeling and Simulation

CFD modeling is a useful tool to gain an additional insight into the physics of the flow and to understand the test results. The objective of the CFD work was to model the flow using 2D CFD to obtain information about the flow that could not be obtained in the experimental test program.

Model Description: The ANSYS FLUENT-14 CFD model was used to model and simulate the laminar flow through a venturimeter. The venturimeter geometry was modeled as a 2D-axisymmetric domain using a structured grid. The dimensions of the geometry were taken from the ISO-5167-1 and no pressure taps were included in the CFD geometry.

The simulation was carried out for a standard classical venturimeter with following dimensions and the model is shown in the Fig.2

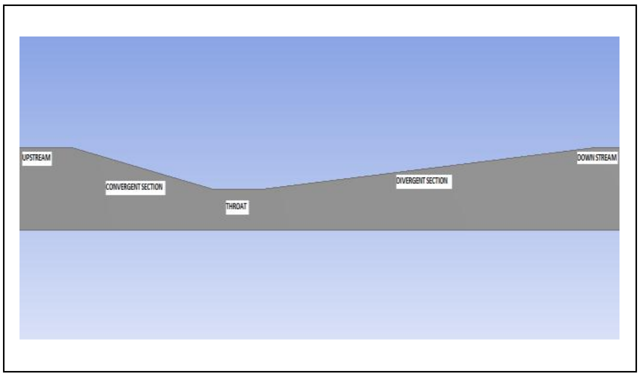


Fig.2. Classical Venturimeter

Variables	Values	Units
Diameter of pipe	100	mm
Diameter of throat	50	mm
β-ratio	0.5	---
Upstream pipe length	6D	mm
Downstream pipe length	20D	mm
Convergent angle	21	deg
Divergent angle	9	deg

Table 1: Dimensions of Venturi geometry

Fig. 3 shows the CFD grid used for the venturimeter simulation. The modeled geometry includes 6D of straight pipe upstream of the convergent section and a 20D of straight pipe downstream of divergent section. The convergent, throat and divergent region were meshed with very fine grids, while the upstream and downstream pipe region was meshed with coarse grids. A total of 183546 quadrilateral elements were used to generate a suitable mesh for the analysis.

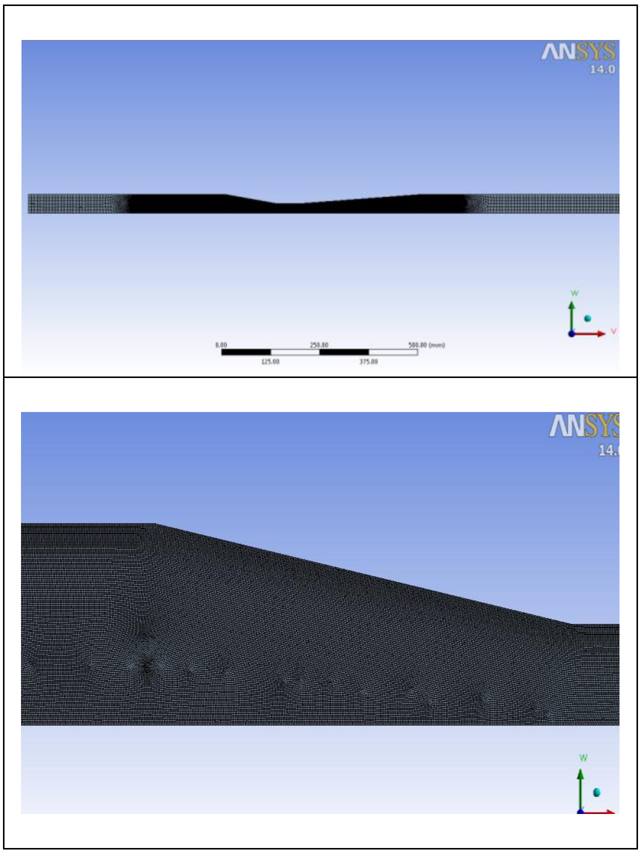


Fig.3. Computational grids used for venturimeter simulation

III. Convergence and Validation of CFD Methodology:

As per the ISO-5167-1 [3], the different types of classical venturimeter based on methods of manufacturing are, classical venturi tube with an "as cast" convergent section, classical venturi tube with a machined convergent section and classical venturi tube with a rough-weld sheet-iron convergent section.

For validation of CFD methodology a standard classical venturimeter with a machined convergent section was selected. As per ISO-5167-1, the standard dimensions of a classical venturimeter with a machined convergent section are in the ranges of,

$50mm < D < 250mm$
$0.4 < \beta < 0.75$
$2x10^5 < Re < 1x10^6$

The ISO-5167-1[3] also gives specification for various geometry details like convergent and divergent angles, length of the throat, sharpness of the edges etc. when the venturimeter is fabricated as per the above requirements and used in the limits of applicability the value of Cd for such a meter would be 0.995 with an uncertainty limit of 1.25%.

Based on the standard limits, a venturimeter with D=100mm and β=0.5 was constructed as a 2D axis symmetric geometry. In the simulation procedure, the process of grid generation is a very crucial step for better accuracy, stability and economy of prediction. Based on mesh convergence study, it is revealed that a total of quadrilateral elements beyond 150000 yields a consistent result. Hence a total of 183546 quadrilateral elements were used.

In the modeling, mass, momentum, energy conservation equations (if necessary) must be satisfied. The governing equations for steady incompressible flows are:

Conservation of mass equation, this equation is valid for both incompressible and compressible flows.

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

Conservation of momentum equation and for steady flow it is written as,

$$\rho\frac{du}{dt}=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$
$$\rho\frac{dv}{dt}=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$
$$\rho\frac{dw}{dt}=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

Where 'p' is the static pressure, 'ρg' is the gravitational body force.

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation.

Among various turbulence models available, the realizable k-ε model is a relatively recent development and differsfrom the standard k- ε model in two important ways:

a) The realizable k-ε model contains a new formulation for the turbulentviscosity.

b) A new transport equation for the dissipation rate, ε, has been derivedfrom an exact equation for the transport of the mean-squarevorticity fluctuation.

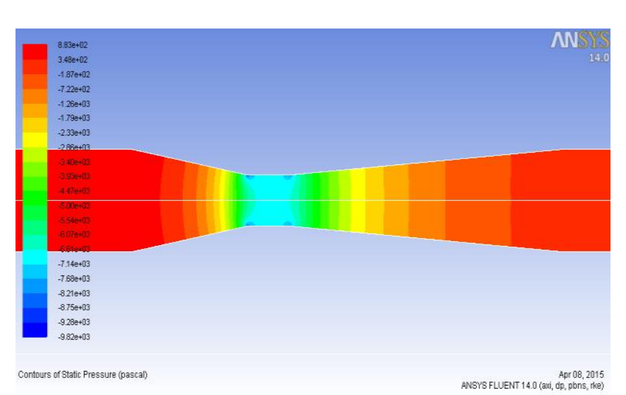
The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG k-ε model is realizable.

An immediate benefit of the realizable k-ε model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

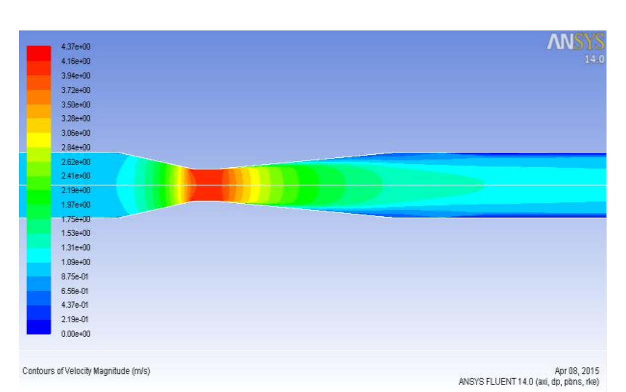
Thus the Realizable k-ε turbulence model with standard wall conditions was selected to model the flow domain as it is superior to the Standard k-ε model for the prediction of separated turbulent flows. SIMPLE algorithm with second order upwind differencing scheme was used for iterative process.Velocity at the inlet was specified as 1m/s and at outlet the gauge pressure was set to zero. The heat transfer from the wall of the domain was neglected.The solution was computed in the commercial CFD code Fluent 14, in which the pressure based solver, was selected for this particular case. The results obtained from the CFD simulations were very close to the C_d value specified in the standard ISO 5167-1 and hence CFD methodology was validated.

Reynolds Number	C_d specified in ISO 5167-1	C_d obtained from CFD
$5x10^5$	0.995	0.989

Table 2.Comparison of Cd value of ISO 5167-1 and CFD Cd value

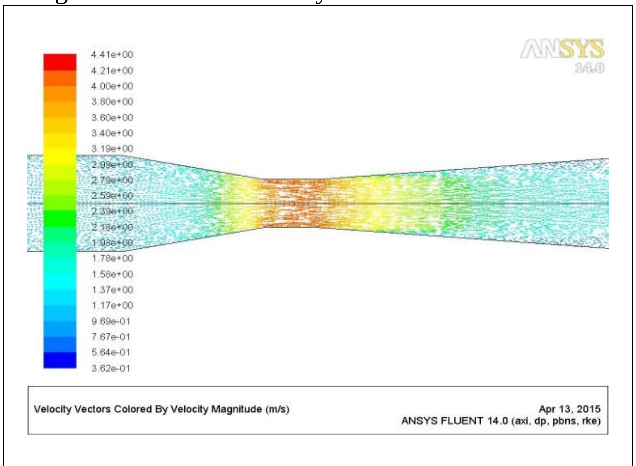


Pressure contour: Max=8.83x10^2, Min= -9.82x10^3



Velocity contour: Max=4.37, Min=0

Fig.4. Pressure and velocity contours of Venturimeter



Velocity vectors: Max=4.41, Min=3.62

Fig.5. Velocity vectors

IV. PREDICTION OF 'C$_D$' AT LOW REYNOLDS NUMBER

In the prediction of C$_d$ at low Re the venturimeter was modeled as per the dimensions specified in the table 2.In simulating a fully-developed laminar flow profile at the Venturi meter inlet, a laminar velocity profile was prescribed at the pipe inlet with throat Reynolds number, located six D upstream of the upstream pressure tap. This ensured a fully-developed laminar velocity profile at the Venturi upstream pressure tap location.

The laminar model with standard wall conditions was selected to model the flowin the laminar region for the prediction of C$_d$. The solution was computed in the commercial CFD code Fluent 14, in which the pressure based solver, was selected for this particular case. The velocity at the inlet was specified as 1m/s, the gauge pressure at the outlet was set to zero Pascal. At the wall no slip condition was specified and the heat transfer from the wall of the domain was neglected. The flow solutions were obtained for steady, incompressible flow. The range of Reynolds number covered in this computation is from 1 to 1500.

Analytical correlation for the discharge coefficient of venturimeter in laminar region:

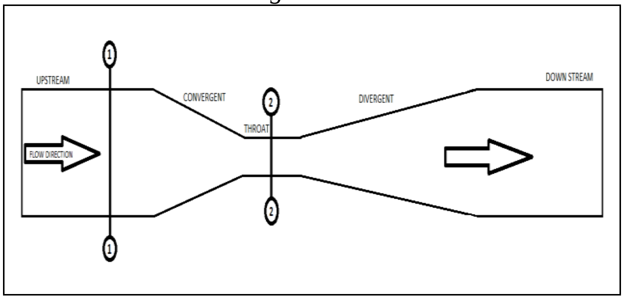


Fig.6. Venturimeter

Assuming the flow is ideal and applying Bernoulli's equation before and after contraction,

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

But Z$_1$= Z$_2$,

$$\frac{(p_1-p_2)}{\rho}=\frac{(v_2^{\,2}-v_1^{\,2})}{2}$$

$$\frac{(p_1-p_2)}{\rho}=\frac{v_2^{\,2}}{2}\left(1-\frac{A_2^2}{A_1^2}\right)$$

$$\frac{\Delta p}{\rho}=\frac{v_2^{\,2}}{2}(1-\beta^4)$$

$$v_2=\frac{1}{\sqrt{1-\beta^4}}\sqrt{\frac{2\Delta p}{\rho}}$$

From continuity equation, we have

$$Q_{th}=A_1v_1=A_2v_2$$

$$Q_{th}=A_2v_2=\frac{1}{\sqrt{1-\beta^4}}\frac{\pi d^2}{4}\sqrt{\frac{2\Delta p}{\rho}}$$

The above equation is based on the assumptions that the flow is steady, incompressible, inviscid, irrotational, no losses and the velocities V$_1$ and V$_2$ are constant across the cross section.

$$Q_{act}=\frac{C_{d\,std}}{\sqrt{1-\beta^4}}\frac{\pi d^2}{4}\sqrt{\frac{2\Delta p}{\rho}} \qquad \text{(A)}$$

The coefficient of discharge 'C$_d$' takes care of real fluid effects i.e., the kinetic energy correction factor, viscous effects and the losses due to sharp edges.

As per ISO-5167-1 [3], for a classical venturimeter the flow with Reynolds number R$_D$> 2x10^5 the value of discharge coefficient 'C$_d$' is equal to 0.995, therefore the losses are only 0.5%. But in case of laminar flow the losses are higher and it is mainly due to viscous losses. When a liquid is flowing through a pipe, the velocity of the liquid layer adjacent to the pipe wall is zero. The velocity of liquid goes on increasing from the wall and thus velocity gradient and hence shear stresses are produced in the whole liquid due to viscosity. This viscous action causes loss of energy which is usually known as frictional loss. Thus the fluid experiences some resistance due to which some of the energy of fluid is lost, this head loss is due to friction and is given by Darcy's law,

$$H_L = \frac{(\Delta p)_{viscous}}{\rho g} = f\frac{v^2}{2g}\frac{L}{D}$$

Therefore,

$$(\Delta p)_{viscous} = f\frac{1}{2}\rho v^2\frac{L}{D}$$

Where, ‘f’ is the friction factor which is a function of Reynolds number and for laminar flow ‘f’ is given by,

$$f = \frac{64}{R_e}$$

Now, applying Bernoulli’s equation between two pressure taps,

$$\frac{p_1}{\rho} + \frac{v_1^2}{2} = \frac{p_2}{\rho} + \frac{v_2^2}{2} + \frac{(\Delta p)_{viscous}}{\rho}$$

$$\frac{(p_1 - p_2)}{\rho} = \frac{(v_2^2 - v_1^2)}{2} + f\frac{1}{2}v_2^2\frac{L}{D}$$

To estimate the value of ‘L/D’ between the section (1) and (2), we will use local ‘D’.

In the pipe, L/D = 0.5

In the throat, L/D = 0.5

In converging portion, for β=0.5

$$\tan\theta = \frac{\left(D/_2 - d/_2\right)}{L_1} = \frac{d}{2L_1}$$

$$\frac{L_1}{d} = \frac{1}{2\tan(10.5)} = 2.7$$

But,

$$d_{avg} = \frac{D + d}{2} = 1.5d$$

Therefore,

$$\left(\frac{L_1}{d}\right)_{act} = \frac{2.7}{1.5} = 1.8$$

Thus the total actual (L/D) between the sections (1) and (2) = (0.5+1.8+0.5) = 2.8

For simplicity, we will put $V=V_2$ in Darcy equation and $f=64/R_{ed}$.

Therefore for laminar flow we get,

$$\frac{\Delta p}{\rho} = \frac{v_2^2}{2}\left[(1 - \beta^4) + f\left(\frac{L_1}{d}\right)_{act}\right]$$

$$\frac{\Delta p}{\rho} = \frac{v_2^2}{2}[(1 - \beta^4) + 2.8f]$$

$$Q_{th} = A_2 v_2 = \frac{1}{\sqrt{[(1 - \beta^4) + 2.8f]}}\frac{\pi d^2}{4}\sqrt{\frac{2\Delta p}{\rho}}$$

$$Q_{act} = \frac{C_{d\,std}}{\sqrt{1 - \beta^4}}\frac{\sqrt{(1 - \beta^4)}}{\sqrt{[(1 - \beta^4) + 2.8f]}}\frac{\pi d^2}{4}\sqrt{\frac{2\Delta p}{\rho}} \quad \text{(B)}$$

Comparing (A) and (B) we get,

$$C_{Laminar} = C_{d\,std}\sqrt{\frac{\sqrt{(1 - \beta^4)}}{[(1 - \beta^4) + 2.8f]}}$$

If β=0.5 then,

$$C_{Laminar} = 0.995\sqrt{\frac{\sqrt{(1 - 0.5^4)}}{[(1 - 0.5^4) + 2.8f]}}$$

$$C_{Laminar} = 0.995\sqrt{\frac{1}{[1 + 3f]}} \quad \text{(C)}$$

Where $f=64/R_{ed}$

Equation (c) gives the discharge coefficient for the laminar flow with Reynolds number less than 2000 and beta ratio equal to 0.5. The above equation does not take into account the corner loses and kinetic energy correction factors. However it should represent the trend in the variation of C_d with Reynolds number in the laminar region.

V. Results and Discussions

CFD simulations were performed over a wide range of conditions. Venturimeter discharge coefficients were calculated from the CFD predicted venturi pressure drops and the flow rates. The simulations were done for different low Reynolds numbers by varying the fluid viscosities and keeping the constant inlet velocity as 1m/s. Initially the simulation was carried out for a standard classical machined venturimeter and the CFD results were validated with the standard.

The simulations for actual low Reynolds number were carried out and the CFD results were validated with the experimental test data available [1]. Additional CFD simulations were conducted to predict Venturi meter operation at Reynolds numbers lower than that could be tested in the laboratory.

The CFD results were also compared with the analytically calculated Cd by using the correlation.

$$C = 0.995\sqrt{\frac{1}{(1 + 3f)}}$$

Where, for laminar flow ‘f’ is given by $f = \frac{64}{R_{ed}}$

The CFD discharge coefficients follow the same general trends as the experimental and analytical results, but CFD results do not agree exactly with any of the correlations. For Reynolds numbers below about 1,000, the data falls within the range of the various published results. Table 3, shows the comparison of CFD results with experimental data available and the analytical result.

In figure 7, the discharge coefficients for the venturimeter from CFD simulations are compared with published data from several sources and also with the analytically predicted discharge coefficients. The discharge coefficients were found to follow the same general trends as the published results. And also the overall behavior of the discharge coefficient curve is found to be consistent with what is described in the literature and ISO 5167-1.

Sl.No.	Re_d	Expt. C_dValue (Stobie et. al [1])	CFD Simulated C_d results	Analytically Calculated C_d Value
1	1	-	0.101	0.072
2	10	-	0.295	0.223
3	35	-	0.460	0.393
4	80	-	0.573	0.542
5	100	0.590	0.602	0.585
6	200	0.700	0.687	0.714
7	500	0.820	0.778	0.850
8	1000	0.880	0.855	0.916
9	1500	0.890	0.875	0.937

Table 3. Comparison of Discharge coefficients

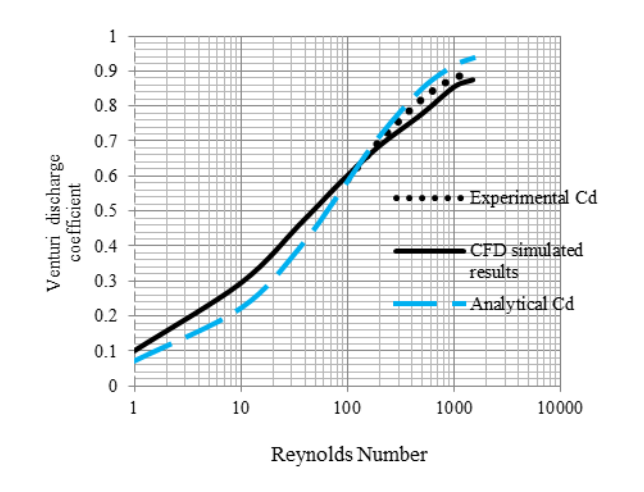


Fig.7: Comparison of published values of venturimeter

discharge coefficients for low Re with the CFD results

The agreement between the analytically calculated and CFD simulated results is good; although it appears that two data sets have slightly different slopes. Also the CFD results were in close comparison with the experimental data [1]. The favorable agreement between the analytical, CFD and experimental results suggests that CFD can be used to model the low Reynolds number flows when using venturimeters.

VI. Conclusions

CFD modeling and simulation was performed to determine if CFD simulations could predict the performance of a venturimeter under non-ISO standard conditions typically encountered in metering viscous fluids. The results obtained from CFD were used to study the detailed information on venturimeter flow characteristics that could not be easily measured during experimental testing at very low Reynolds number.

Considering the uncertainty involved in the experimental study and CFD analysis, the agreement between the computed values and experimental data of C_d can be considered as satisfactory.It can also be inferred that for a laminar flow in a specific venturimeter (fixed D and β), the discharge coefficient is a function of throat Reynolds number over a range of viscosities and flow rates.

It is observed that as the Reynolds number decreases, the value of C_d reduces very rapidly. This can be attributed to the viscous losses that occur at these Reynolds number. Hence while using venturimeter at low Reynolds number, it is essential to use the correct value of C_d.

It is also necessary to be mentioned that since C_d varies significantly with Re (hence flow rate), it is not very convenient to use venturimeter for such applications because Cdwill not be known a prior. Hence it would require an iteration process to arrive at the actual flow rate.

References

[1]Gordon Stobie, Conoco Phillips, Robert Hart and Steve Svedeman, Southwest Research Institute. *Erosion in a Venturi Meter with Laminar and Turbulent Flow and Low Reynolds Number Discharge Coefficient Measurements.*

[2]C.L. Hollingshead, M.C. Johnson, S.L. Barfuss, R.E. Spall (2011*). Discharge coefficient performance of Venturi, standard concentric orifice plate, V-cone and wedge flow meters at low Reynolds numbers.* Journal of Petroleum Science and Engineering.

[3]Indian standard ISO-5167-1, *Measurement of fluid flow by means of pressure differential devices* (1991).

[4]T. Nithin, Nikhil Jain and AdarshaHiriyannaiah (2012). *Optimization of Venturi Flow Meter Model for the Angle of Divergence with Minimal Pressure Drop by Computational Fluid Dynamics Method*. International Conference on Challenges and Opportunities in Mechanical Engineering, Industrial Engineering and Management Studies.

[5]Nikhil Tamhankar, Amar Pandhare, AshwinkumarJoglekar, VaibhavBansode (2014). *Experimental and CFD analysis of flow through venturimeter to determine the coefficient of discharge*. International Journal of Latest Trends in Engineering and Technology.

[6] Naveenji Arun, Malavarayan S, Kaushik M, H.Sriananth (2010). *CFD analysis on discharge coefficient during non Newtonian flows through orifice meter.* International Journal of Engineering Science and Technology Vol. 2(7).

[7] P. Hari Vijay, V. Subrahmanyam(2014). *CFD simulation on different geometries of venturimeter*. International Journal of Latest Trends in Engineering and Technology vol-3