Analysis of Flow Through Rotameter Using CFD

Pavan Kumar K P¹, Dr. V Seshadri², Puneeth Kumar S B³
¹M.Tech Student, Thermal Power Engineering, MIT-Mysore, India
²Professor (Emeritus), ³Assistant Professor, Department of Mechanical Engineering, MIT-Mysore, India

Abstract—The objective of the present work is to analyze the flow through Rotameter using CFD tool ANSYS FLUENT 14 so that design can be made on this basis. Here a rotameter available in the fluid mechanics laboratory of the Institute, having a range of 4–40 lpm for water flow is considered for the analysis. All the Dimensions of the tapered tube and float were measured accurately and the flow domain is modeled using ANSYS design modeler. The flow domain is discretized using quadrilateral elements. The drag force on the float at various positions in the tapering tube has been calculated. On the basis of the analysis the flow rates at which the float comes to equilibrium at various locations have been calculated. A comparison between the computed (CFD) flow rate and the flow rate indicated by the meter shows an excellent agreement. The study has demonstrated that CFD can be used for the design of rotameter as long as sufficient care is taken to ensure proper discretization and choice of turbulence model.

Keywords—Rotameter, Drag force, ANSYS, Turbulent flow, CFD

I. INTRODUCTION

Flow meter is a device that measures the rate of flow or quantity of a moving fluid in an open or closed conduit. There are large varieties of flow meters that are used in the industry for different applications. Some of the major categories of metering devices are obstruction type (like venturimeter, orifice meter, nozzle etc.), positive displacement flow meters, turbine flow meters, ultrasonic flow meters etc.

Rotameter is a Variable area type of flow meter. It consists of a tapered glass tube and a metallic float which is free to move up and down within the tube. The metering tube is mounted vertically with the smaller end at the bottom as shown in Fig.1. The fluid whose flow rate to be measured enters at the bottom of the tube, passes upward around the float (annulus area), and exists out from the top. When there is no flow through the meter, the float rests at the bottom of the metering tube where the maximum diameter of the float is approximately same as the bore of the tube. When fluid enters the metering tube, the buoyant effect of the fluid lightens the float, but it has a greater density than the liquid and the buoyant effect is not sufficient to raise it. There is a small annular opening between the float and tube. The pressure drop across the float increases and raises the float. With upward movement of float towards the larger end of the tapered tube, the annular opening between tube and the float increases. As area increases, the pressure differential across the float decreases. The float assumes a position when it is in dynamic equilibrium. This happens when the drag force due to the fluid flow on the float plus the buoyancy effect balances the weight of the float. Any further increase in flow rate causes the float to rise higher in the tube; a decrease in flow causes the float to drop at a lower position. Every float position corresponds to one particular flow rate. Graduated markings are provided for different flow rates on a calibration scale on the outer surface of the tube and flow rate can be determined by direct observation of the position of float in the metering tube.

The objective of the present study is to analyze the flow through rotameter using CFD so that the design of rotameter can be made on this basis. Here, by considering the dimensions of the rotameter available in the fluid mechanics lab for a flow range of 4–40 lpm. A 2D axisymmetric model of rotameter is created using ANSYS Design Modeler based on available dimensions. For each flow rate and the position of the float, the value of drag force is obtained using CFD. The value of flow rate at each position where the float is in dynamic equilibrium is calculated. Thus, the calculated flow rate at each position where the float is in dynamic equilibrium is calculated. Thus, the calculated flow rate using CFD and indicated flow rate are compared.
II. PRINCIPLE OF ROTAMETER

A rotameter test rig considered for the study is shown in Fig.1. The various forces that are acting on the float at any given position are as follows:

1. The weight of the float (W) which is acting vertically downwards
2. The buoyant force (F_B) acting vertically upwards
3. The drag force (F_D) due to fluid flow.

When the above 3 forces are balanced the float comes to equilibrium. It is to be noted that the first two forces namely W and F_B are constant for a given meter. However, the drag force F_D is dependent on several parameters like the properties of the fluid, flow rate, position of the float etc. Normally the drag force consists of pressure drag (form drag) and viscous drag. Since the float is a bluff body, the pressure drag dominates the viscous drag. However the drag force on the float is also dependent on the extent of annulus area available for flow between the float and divergent tube. Thus computation of F_D and its dependence will be very complex.

The various forces acting on the float can be calculated as follows:

Force due to the weight of the float acting downwards due to gravity (W) is given by:

\[ W = V_S \cdot \rho_f \cdot g \] ........................ (1)

Where, \( V_S \) is the total volume of float in m³
\( \rho_f \) is the density of the float material in Kg/m³
\( g \) is the gravitational force in m²/s

Force applied by the fluid which is acting upwards due to buoyancy effect (F_B) is given by:

\[ F_B = V_S \cdot \rho_f \cdot g \] ........................ (2)

Where, \( \rho_f \) density of fluid whose flow rate is measured

Drag force on the float due to the flow of fluid from inlet, which is acting upwards is given by:

\[ F_D = \frac{C_D \cdot A_S \cdot \rho_f \cdot U_a^2}{2} \] ........................ (3)

Where, \( C_D \) = Drag coefficient which is a dimensionless number
\( A_S \) = projected cross sectional area of the float.

This Drag coefficient has been defined with respect to average velocity of the fluid in the annulus area between float and the tapering tube (U_a).

When the flow starts, the float will rise from its initial position due to these forces and reaches a position where all the forces acting on it are balanced. The equilibrium state is reached when

\[ F_D = W - F_B \]

The flow rate (Q) is given by continuity equation:

\[ Q = (U_m \times A_m) = (U_A \times A_s) \]

\[ Q = (U_m \times A_m) = \left[ \frac{\pi}{4} \times (D_t^2 - D_s^2) \right] m^3/s \] ............ (4)

Where, \( U_m \) is the average velocity at the inlet of the tube, m/s
\( A_m \) is the area at the inlet of the tube, m²
\( D_s \) is the maximum diameter of the float, m
\( D_m \) is the inlet diameter of the tube, m
\( D_t \) is the diameter of the tube at the flow rate markings or float position with respect to the flow rate, m
\( A_s \) is annulus area between the float and the tapered tube, m²
\( U_A \) is the average velocity measured at the annulus area between the tapered tube and the float at different locations with respect to flow rate (Q) and it is given by,

\[ U_A = \frac{\sqrt{2 \cdot (\rho_s - \rho_l) \cdot V_S \cdot g}}{\rho_f \cdot C_D \cdot A_S} \text{ m/s} \] .......... (5)

Considering the above equations, it is possible to form the general flow equation for variable area flow meters.

From equations (4) and (5) we get Volume rate of flow:

\[ Q = \frac{\pi}{4} \times (D_t^2 - D_s^2) \times \frac{2 \cdot (\rho_s - \rho_l) \cdot V_S \cdot g}{\rho_f \cdot C_D \cdot A_S} \text{ m}^3/\text{s} \] .......... (6)
The half divergence angle of the tapering tube ($\theta$) is given by

$$\tan \theta = \frac{D_{in} - D_{out}}{2L}$$

$D_{out}$ is the diameter of the tube at the outlet, m

$$(D_t^2 - D_S^2) = [4.H.D_s\tan \theta + 4.H^2. (\tan \theta)^2] \ldots (7)$$

Substituting Equation (7) in (6) we get relation between height of the float and volumetric flow rate provided the value of $C_D$ and geometric dimensions of the rotameter are known. From this the value of drag coefficient can be obtained.

It is observed from the literatures that analysis of flow through an actual rotameter has not been done with a purpose of comparing the computed flow rate with the indicate flow rate to assess the accuracy of the flow rate. The scope of the present study is to analyze the flow through rotameter using CFD and comparing the results with actual experimental results. It is expected that CFD can be used as a design tool for rotameters thereby replacing the actual calibration.

### III. CFD METHODOLOGY AND VALIDATION

The flow through rotameter is assumed to be that of an incompressible Newtonian fluid. Thus the governing equations that need to be solved are the continuity and Navier Stokes equations with proper boundary conditions. However if the flow is turbulent, Reynolds Averaged Navier Stokes equations (RANS) need to be solved and additional equations for turbulent quantities have to be included in the analysis. This is the process of turbulence modeling. The above equations are a set of coupled nonlinear partial differential equations. Hence the CFD software uses numerical techniques like finite volume method (FVM) to solve these equations. The description of the governing equations, solution methodology and techniques are given in standard literature [7] and hence they are not mentioned here. However, it is very essential that before solving any flow, the methodology has to be validated by analyzing a flow problem for which solution is already available. Such a validation will help in ensuring convergence and proper choice of turbulence model. In the present study, the developing flow through a circular pipe in turbulent regime is chosen for validation. The flow domain is shown in Fig.2.

The various dimensions of the pipe analyzed are as follows

- Length of the pipe (L) = 7000mm
- Diameter of the pipe (D) = 100mm.

Boundary conditions applied are a constant inlet velocity $= 1$ m/s, wall condition at the pipe wall, zero gauge pressure at the outlet and axis along the center line of the pipe. The fluid density is taken as 1000 kg/m$^3$ and viscosity $\mu = 0.01$ Pa-s so that Reynolds number of the flow is $10^4$. Two Dimensional axisymmetric circular pipe model has been modeled using ANSYS Design Modeler. According to the mesh convergence study the mesh having approximately 40000 quadrilateral elements yielded accurate results. Different turbulence models like Standard $k-\epsilon$, RNG $k-\epsilon$, Realizable $k-\epsilon$, $k-\omega$ Standard and $k-\omega$ SST have been studied. The study revealed that $k-\omega$ SST model gives closest results that compare with standard solutions available [6]. The development length $L_e$ is defined as the distance from the inlet at which the axial velocity reaches 99% of the fully developed value. As per the standard literature this length is given by

$$L_e = (4.4 \times Re^{0.167} \times D) \text{ m}$$

And the ratio of $U_{max}/U_{avg}$ at this Reynolds number is 1.25,

Further the pressure drop in the fully developed region is given by Darcy- Weishback equation [6]

$$\Delta P = \frac{f \times \rho \times U_{avg}^2 \times L}{D} \text{ Pa}$$

Where, $f =$Friction factor $= \frac{0.3264}{Re^{0.25}}$ for smooth pipes, $\rho =$density of fluid, Kg/m$^3$  
$L =$ distance between two locations, m  
$D =$ Diameter of the pipe, m

The results obtained are given in Table.I and Figs 3-6
Fig.3 shows the variation of axial velocity along the length of the pipe. It is observed that as the flow develops, the axial velocity increases and then reaches a fully developed value which is found to be (1.243U_in) which compares very favorably with the standard value (see Table.I). Further the computed distance at which this value is reached (L_e) also agrees with standard value. The computed values of pressure drop in the pipe between two locations at a distance of 4m and 5m respectively from the inlet have been calculated. The flow is fully developed in this region. As seen from the Table.1 the computed and standard values of pressure drop agree very well. Fig.4 shows the vector plots in the inlet region of the pipe where the flow is developing. They depict the acceleration of the fluid near the axis clearly. Fig.5 shows the pressure contour near the inlet of the pipe. Fig.6 shows the variation of pressure along the length of the pipe which demonstrates that pressure gradient becomes constant when once the flow is fully developed. The results tabulated in the Table.3 have established that the convergence is satisfactory and choice of turbulence model is proper. Thus the methodology is taken as validated.

IV. CFD MODELING AND SIMULATION

For the purpose of analysis, the flow through a rotameter existing in the fluid mechanics laboratory of the Institute is chosen. As a first step the dimensions of the rotameter are measured accurately. The specifications of the rotameter and various dimensions are as follows.

A. Details of the Rotameter Analyzed

The rotameter is for the measurement of flow rate of water in the range 0 – 40 LPM. It consists of a vertical diverging tube (see Fig 7) in which a steel float can move freely. The shape of the steel float is shown in Fig.8. The various dimension details are as follows

<table>
<thead>
<tr>
<th>Parameters</th>
<th>CFD results</th>
<th>Standard results</th>
</tr>
</thead>
<tbody>
<tr>
<td>ΔP (Pa)</td>
<td>164</td>
<td>163</td>
</tr>
<tr>
<td>U_{max}</td>
<td>1.243</td>
<td>1.25</td>
</tr>
<tr>
<td>U_{avg}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>L_e</td>
<td>20.5</td>
<td>20.42</td>
</tr>
</tbody>
</table>

Fig.4: Velocity vector of developing turbulent flow

Fig.5: Pressure contour near the inlet of the pipe

Fig.6: Variation of pressure along the length of the pipe
1) Dimensions of the Tapered Tube:

Inlet diameter = \( D_{in} = 25.48 \) mm
Outlet diameter = \( D_{out} = 36.85 \) mm
Vertical height of the tube = \( L = 260 \) mm
Half Taper angle = \( \theta = 1^{\circ} 12^{\prime} \).

2) Dimensions of the float:

Float Material = stainless steel
Material Density = 7850 kg/m\(^3\)
Volume of float = \( V_s = 12584.46 \) mm\(^3\)

Drag force \( F_D \) when the float is in equilibrium is calculated using the formulae,

\[
F_D = (\rho_{s} - \rho_f) \times V_s \times g \quad \text{Newton} \\
= (7850 - 1000) \times 12584.46 \times 10^{-9} \times 9.81 \\
= 0.8456 \text{ N}
\]

This \( F_D \) is constant at all positions when the float is in equilibrium since it depends only on density of float material and fluid and volume of the float.

B. Flow rate markings

Flow rate markings have been made on the glass tube at various intervals in the range 4-40 lpm. The height of each marking from the inlet has been measured and are tabulated in Table.II

<table>
<thead>
<tr>
<th>Flow rate(Q) lpm</th>
<th>H (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>57.5</td>
</tr>
<tr>
<td>7</td>
<td>72.5</td>
</tr>
<tr>
<td>10</td>
<td>86.5</td>
</tr>
<tr>
<td>15</td>
<td>108.5</td>
</tr>
<tr>
<td>20</td>
<td>129</td>
</tr>
<tr>
<td>25</td>
<td>148</td>
</tr>
<tr>
<td>30</td>
<td>166</td>
</tr>
<tr>
<td>35</td>
<td>183</td>
</tr>
<tr>
<td>40</td>
<td>199</td>
</tr>
</tbody>
</table>

V. Flow domain and boundary conditions used in CFD modelling

Fig.9 shows the flow domain used in the CFD analysis. It is identical to the rotameter described earlier. The geometry considered is axisymmetric and hence only half the domain is shown. The inlet and outlet diameters as well as the length of the tube are taken corresponding to values of the actual rotameter.

With these available dimensions the 2D axisymmetric rotameter model is constructed using ANSYS Design Modeler. In the stimulation process, the grid generation plays a vital role for better accuracy and stability in the prediction of results. Since the area around the float increases as the float height increases, the mesh convergence study is done only for one particular flow rate i.e., Q=4lpm and height of the float (H) = 57.5mm corresponding to that flow rate. The boundary conditions applied are inlet velocity = 0.12998 m/s, the glass tube and the float surface are considered as wall, and the centerline as axis. The simulation is done for different mesh sizing. According to the mesh convergence study, it observed that for quadrilateral elements more than 85000, the computed values of drag force is accurate and remains constant. Different turbulence models have been studied and it has been concluded that k-ω, SST model yields more accurate results than other models.

Fig.10 shows the mesh used around the float region which has been made very fine since the velocity gradients are expected to be large in this region. Figs.11 and 12 show the velocity contours and velocity vector plots in the region around the float respectively. They clearly show the separation of flow and velocity variation in the region. Fig 13 shows the velocity vector behind the float. It clearly shows the reversed flow taking place as the flow passes the annulus region due to the flow separation. The vortex formation can be seen in this region. The pressure contours shown in Fig.14 also describe the variation of pressure as the flow passes over the float.

The final computations have been made with float in different positions corresponding to various flow rates (see Table.III). The fluid is assumed to be water with density = $10^3$ Kg/m$^3$ and viscosity = 1Cp. Boundary conditions on the glass tube and float surface is wall and axis boundary condition is given on the centerline (see Fig.9) Zero gauge pressure is specified at the outlet. Secondary upwind scheme is used for solution.
TABLE III: Flow Rate and Corresponding Inlet Velocity

<table>
<thead>
<tr>
<th>Flow rate (Q_{ind}) in LPM</th>
<th>Inlet velocity(U_{in}) in m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>0.12998</td>
</tr>
<tr>
<td>7</td>
<td>0.2270</td>
</tr>
<tr>
<td>10</td>
<td>0.3250</td>
</tr>
<tr>
<td>15</td>
<td>0.4860</td>
</tr>
<tr>
<td>20</td>
<td>0.6490</td>
</tr>
<tr>
<td>25</td>
<td>0.8107</td>
</tr>
<tr>
<td>30</td>
<td>0.9750</td>
</tr>
<tr>
<td>35</td>
<td>1.1350</td>
</tr>
<tr>
<td>40</td>
<td>0.12998</td>
</tr>
</tbody>
</table>

Where, Q_{ind} is the flow rate indicated on the rotameter. Computations have been made for 9 locations of the float corresponding to different flow rates as given in Table.IV. At each location, the inlet velocity corresponding to that location is specified and the drag force on the float is computed. The calculated values of drag force at various positions of the float are given in Table.V. It is observed that although the computed values of drag force are close to the value required for the equilibrium (0.8456 N) they are not exactly equal. In such cases, the inlet velocity is changed accordingly so that drag force achieves this value. Then the flow rate corresponding to that velocity is calculated. The final results obtained are shown in Table.VI. The percentage errors between the indicated flow rate and the flow rate obtained from CFD are tabulated in Table.VII.

The maximum percentage error is seen for the flow rate Q_{ind} = 10 lpm which is 1.66%. At other flow rates the deviations are much smaller. Fig.13 shows the plot of Q_{ind} Vs Q_{CFD}. From the graph the deviations between indicated flow rate and computed flow rates are very small. Almost all the data points are very close to best fit line.

TABLE IV: Position of Float (H) for Different Flow Rates (Q_{ind})

<table>
<thead>
<tr>
<th>Flow rate(Q_{ind}) lpm</th>
<th>H (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>57.5</td>
</tr>
<tr>
<td>7</td>
<td>72.5</td>
</tr>
<tr>
<td>10</td>
<td>86.5</td>
</tr>
<tr>
<td>15</td>
<td>108.5</td>
</tr>
<tr>
<td>20</td>
<td>129</td>
</tr>
<tr>
<td>25</td>
<td>148</td>
</tr>
<tr>
<td>30</td>
<td>166</td>
</tr>
<tr>
<td>35</td>
<td>183</td>
</tr>
<tr>
<td>40</td>
<td>199</td>
</tr>
</tbody>
</table>

TABLE V: Computed Values of Drag Force (F_{D}) at Various Q_{ind} and H

<table>
<thead>
<tr>
<th>Q_{ind}(lpm)</th>
<th>H (mm)</th>
<th>U_{in} (m/s)</th>
<th>F_{D} (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>57.5</td>
<td>0.12998</td>
<td>0.8463</td>
</tr>
<tr>
<td>7</td>
<td>72.5</td>
<td>0.2270</td>
<td>0.8369</td>
</tr>
<tr>
<td>10</td>
<td>86.5</td>
<td>0.3250</td>
<td>0.8238</td>
</tr>
<tr>
<td>15</td>
<td>108.5</td>
<td>0.4860</td>
<td>0.8356</td>
</tr>
<tr>
<td>20</td>
<td>129</td>
<td>0.6490</td>
<td>0.8369</td>
</tr>
<tr>
<td>25</td>
<td>148</td>
<td>0.8107</td>
<td>0.8479</td>
</tr>
<tr>
<td>30</td>
<td>166</td>
<td>0.9750</td>
<td>0.8480</td>
</tr>
<tr>
<td>35</td>
<td>183</td>
<td>1.1350</td>
<td>0.8466</td>
</tr>
<tr>
<td>40</td>
<td>199</td>
<td>1.2998</td>
<td>0.8553</td>
</tr>
</tbody>
</table>

TABLE VI: Computed Values of the Flow Rate Obtained from CFD at Various Indicated Flow Rates

<table>
<thead>
<tr>
<th>Q_{ind} (lpm)</th>
<th>U_{CFD} (m/s)</th>
<th>Q_{CFD} (lpm)</th>
<th>F_{D} (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>1.2992</td>
<td>4.006</td>
<td>0.8456</td>
</tr>
<tr>
<td>7</td>
<td>0.2282</td>
<td>7.035</td>
<td>0.8455</td>
</tr>
<tr>
<td>10</td>
<td>0.3297</td>
<td>10.166</td>
<td>0.8453</td>
</tr>
<tr>
<td>15</td>
<td>0.4891</td>
<td>15.075</td>
<td>0.8455</td>
</tr>
<tr>
<td>20</td>
<td>0.6526</td>
<td>20.12</td>
<td>0.8456</td>
</tr>
<tr>
<td>25</td>
<td>0.8087</td>
<td>24.94</td>
<td>0.8456</td>
</tr>
<tr>
<td>30</td>
<td>0.9735</td>
<td>30.02</td>
<td>0.8457</td>
</tr>
<tr>
<td>35</td>
<td>1.13435</td>
<td>34.976</td>
<td>0.8457</td>
</tr>
<tr>
<td>40</td>
<td>1.29194</td>
<td>39.835</td>
<td>0.8457</td>
</tr>
</tbody>
</table>

Fig.13: Plot of Q_{ind} Vs Q_{CFD} (The solid line represents the best fit line corresponding to a slope of unity)
VI. CONCLUSIONS

From this study it is observed that the values of flow rate computed from CFD are in excellent agreement with that of the flow rate indicated on the metering tube with minor errors. Hence it can be concluded that the CFD methodology can be used to accurately predict the performance characteristics of a rotameter and hence the design can be based on such analysis. The study is being extended to quantify the effect of fluid density and viscosity on the rotameter accuracy.

When once the methodology has been established the calibration of the rotameter with different fluids can be avoided.

REFERENCE


