

**INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH
TECHNOLOGY****DESIGN ANALYSIS FOR A ROTAMETER USING CFD****Deepu C N*, Yogesh Kumar K J, Dr V Seshadri**

*M.Tech Student, Thermal Power Engineering, Maharaja Institute of Technology, Mysore
Assistant Professor, Department of Mechanical Engineering, Maharaja Institute of Technology, Mysore
Professor (Emeritus), Department of Mechanical Engineering, Maharaja Institute of Technology, Mysore

DOI: 10.5281/zenodo.57043

ABSTRACT

Rotameter is a variable area type of flow meter used to measure the flow rate of liquids and gases. In the present study design and analysis of Rotameter performance is done using ANSYS FLUENT 14 software. Validation of CFD methodology is done by using flow through pipe in a turbulent flow. K- ω -SST model was found to be best suited for this class of flows. In the design of Rotameter assuming C_D as 2.4 and maximum flow rate of 100 lpm, preliminary design of a Rotameter has been done for the flow of water. After simulation using CFD flow rate obtained from CFD is not same as that from the preliminary designed values. Hence it is concluded that coefficient of drag is also dependent on diameter ratio. Improved design methodology of Rotameter is presented by considering effect d/D on coefficient of drag, by using a correlation derived on the basis of CFD computations.

The effect of upstream disturbance (like valves) at various locations on the performance of Rotameter is analyzed using validated CFD methodology. It is concluded that as long as the locations of disturbance is more than $5D_{in}$ upstream of the inlet, the error in the measurement will be less than 2%. The validated CFD methodology is used to analyse the effect of density and viscosity of fluid on the accuracy of metering by a Rotameter. A need for the development of correlation for viscosity correction factors are highlighted. The analysis also demonstrated that a Rotameter designed and calibrated for water flow can be used for measurement of air flow at different pressures as long as correction factor are applied. The maximum error of in such cases will be order of 5%. The study has demonstrated that a validated CFD methodology can be used to accurately predict the performance of Rotameter even under non-standard conditions

KEYWORDS: Rotameter, Coefficient of Drag, Drag Force, flow rate.**INTRODUCTION**

Flow meter is a device which measures the flow rate of any liquid in a closed or open conduit. Flow metering is the one of the features in most of the industrial process. A wide variety of flow meters are used for different applications. Some of these devices meters are Obstruction type flow meters like orifice meter, venturi meter and flow nozzles etc. as well as special meters like turbine flow meter, ultrasonic flow meter and electro-magnetic flow meter etc. Specific design like Coriolis meter, Doppler velocity meter etc. Rotameter is one kind of flow meter which is a variable area type of flow meter. It consists of vertically placed glass tapered tube and metallic float free to move up and down within the tube. The metering tube placed vertically along with fluid stream with a larger diameter at the top, fluid enters at the bottom of the tube and passes over the float and leaves at the top.

The maximum diameter of float is approximately same as the inlet diameter of the tube, when fluid enters at the bottom of tube float start rises in the tube. When there is no flow in the tube float is settled at the bottom of metering tube, fluid enters at the bottom of the tube the buoyant effect will act on the float but float density higher than the fluid, position of float still in the bottom of tube. Increasing the flow rate of fluid until buoyant effect of fluid greater than weight of float and then float starts to rise in the tube, it can be move up to top end of tapered tube. When buoyant force and drag force of the fluid on the float is same as that of weight of float, then the float is in dynamic

equilibrium. This position of float corresponds to the particular flow rate of the fluid. If there is increase in the flow rate of fluid float rises upward in the tube and decrease in the flow rate causes drop in height of float in the tube. So flow rate of the fluid can be measured in the Rotameter by direct reading the position of float in the metering tube.

LITERATURE REVIEW

Pavankumar et.al [1&2] conducted experiments in CFD tool ANSYS FLUENT 14 software to analyze flow through a Rotameter, that was available in Fluid Mechanics Laboratory of their Institute. Here the geometry was considered as the 2D axisymmetric and meshing of geometry done by quadrilateral. Water is considered as fluid and steel is used as the float. From the experiments it is concluded that use of CFD gives good accuracy results within $\pm 1\%$ errors and the drag force will remain same from 4 – 40 lpm flow rate of water.

Rakesh Joshi et.al[3] conducted experiments in a Gambit and ANSYS Fluent software to study the effect of Rotameter floats design on the performance of a Rotameter. From the experiments it is concluded that position of float remains same for the small change in flow rate of water and performance of Rotameter directly depends on the float design and drag force offered on the float.

Saumil B Patel [4] used the CFD software for the analysis of an aerofoil, to find the coefficient of drag and coefficient of lift. Here NACA 0012 aerofoil structure is used and air is used as fluid. The experiments are done for two cases one for zero degree of angle of attack (AOA) and six degree angle of attack. From the experiments it shows that at zero degree of angle of attack there is no lift force but there will be drag force and at six degree angle of attack there will be a lift force of 888.72N and more drag. Finally it is concluded that as the angle of attack increases the lift force will increase and more drag force will be acting on it.

Sapna and Abdul Hakeem [5] conducted experiments in COMSOL multi physics software to study the measurement of flow rate of conducting fluids by non -contact, non - intrusive method. Here Lorentz force volumetry (LFV) method was used to measure the drag force and flow rate by providing high magnetic field over the tube through which fluid is flowing. It is concluded that the drag force increases with mass of magnet system and flow rate is calculated using electromagnetic force.

PRINCIPLE OF ROTAMETER

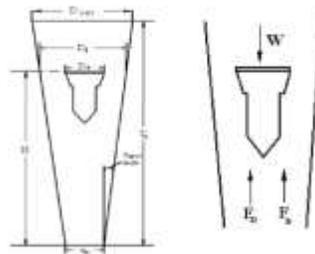


Fig 1 Geometry of Rotameter and Various Forces Acting on the Float

The above figure shows the geometry of tapered tube and float within it and The various forces acting on the float. out of three forces acting on the float value of weight of float (W) and buoyant force (F_B) of the fluid is constant, the value of drag force (F_D) is dependent on the shape of float, velocity of the fluid, position of float in the tube and annulus area between the float and tube.

When the all three forces in equilibrium position then equation becomes

$$F_D = W - F_B \dots\dots\dots (1)$$

Where

F_D= Drag force of the fluid acting on the float vertically upwards

W=weight of the float acting vertically downwards

F_B =buoyant force of fluid acting vertically upwards

Weight of the float acting vertically downward is given by

$$W = V_s \cdot \rho_s \cdot g \dots\dots\dots (2)$$

Where ρ_s =density of steel float

V_s= volume of the float

g = acceleration due to gravity

Buoyancy force of fluid acting on the float is given by

$$F_B = V_s \cdot \rho_f \cdot g \dots\dots\dots (3)$$

Where ρ_f = density of fluid

Drag force of the fluid acting on the float is given by

$$F_D = 0.5 \cdot C_D \cdot \rho_f \cdot U_{anu}^2 \cdot A_s \dots\dots\dots (4)$$

Substitute equation (4)and (3) in equation

$$\begin{aligned} F_D &= W - F_B = (\rho_s - \rho_f)g \cdot V_s \\ &= 0.5 \cdot C_D \cdot \rho_f \cdot U_{anu}^2 \cdot A_s \dots\dots\dots (5) \end{aligned}$$

Flow rate of the fluid in the inlet of the tube and annulus region is given by the continuity equation is

$$\begin{aligned} Q &= (U_{in} \times A_{in}) = (U_{anu} \times A_{anu}) \\ &= \frac{\pi}{4} (D_t^2 - D_f^2) \times U_{anu} \text{ m}^3/\text{s} \dots\dots\dots (6) \end{aligned}$$

D_t =Diameter of the tube when float is in equilibrium position.

D_f= maximum Diameter of the float

A_{in}= inlet c/s area of the tube

A_{anu}= Annulus Area between the float and tube when float is in equilibrium

From the Eqn (4)A_{anu} can also be written as

$$U_{anu} = \sqrt{\frac{2 (\rho_s - \rho_f)g \cdot V_s}{\rho_f C_D \cdot A_s}} \dots\dots \text{m/s} \dots\dots (7)$$

Substituting Eqn (7) in Eqn (6) we get

$$Q = \frac{\pi}{4} \times (D_t^2 - D_f^2) \times \sqrt{\frac{2 (\rho_s - \rho_f)g \cdot V_s}{\rho_f C_D \cdot A_s}} \text{ m}^3/\text{s} \dots\dots (8)$$

The half divergence angle of the tapered tube is given by

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

Where D_{in} and D_{out} - inlet and outlet diameter of the tube T_L - vertical height of the tube

Consider the equation for the tapered tube from the trigonometric relation

$$\begin{aligned} D_t &= D_f + 2 H \tan \theta \\ (D_t^2 - D_f^2) &= 4 \cdot H \cdot D_s \cdot \tan \theta + 4 \cdot H^2 \cdot (\tan \theta)^2 \dots\dots\dots (10) \end{aligned}$$

Substituting Eqn (10) in Eqn (8) we get

$$Q = \frac{\pi}{4} [4 \cdot H \cdot D_s \cdot \tan \theta + 4 \cdot H^2 \cdot (\tan \theta)^2] \times \sqrt{\frac{2 (\rho_s - \rho_f)g \cdot V_s}{\rho_f C_D \cdot A_s}} \text{ m}^3/\text{s} \dots\dots (11)$$

The above relation gives the between the flow rate of fluid, position of float of any Rotameter

Density Correction Factor

Rotameter is also used to measure the flow rate of different fluids by changing the density and viscosity of the fluid. Changes in the density of fluid will effect flow rate of the measured value significantly, but changes in viscosity of fluid will not much more effect on the flow rate. Hence Rotameter need a density correction factor to measure the flow rate of different fluids.

The flow rate of the water in the Rotameter from the Eqn (11) given by

$$Q_w = k \cdot \sqrt{(\rho_s - \rho_w) / \rho_w} \cdot h \quad m^3/s \quad \dots\dots (12)$$

Where k is constant term

ρ_s = density of the steel float

ρ_w = density of water

h= height of the float

Suppose same Rotameter used to measure flow rate of any fluid, then adjusted the flow rate until float reaches same height of h

$$Q_L = K \cdot \sqrt{(\rho_s - \rho_L) / \rho_L} \cdot h \quad m^3/s \quad \dots\dots (13)$$

ρ_L = density of any liquid

Divide Eqn (13) by Eqn (12), we get

$$\frac{Q_L}{Q_w} = \sqrt{\frac{(\rho_s - \rho_L) \rho_w}{(\rho_s - \rho_w) \rho_L}} \times Q_w \quad m^3/s \dots\dots (14)$$

Therefore $Q_L = C_s \cdot Q_w \quad \dots\dots (15)$

Where

C_s = density correction factor)

$$C_s = \sqrt{\frac{(\rho_s - \rho_L) \rho_w}{(\rho_s - \rho_w) \rho_L}} \quad \dots(16)$$

CFD METHODOLOGY AND VALIDATION

Methodology of CFD involves in the flow through Rotameter is assumed as Newtonian, incompressible and two dimensional fluid flow. Governing equations are used to solve this problem are ,conservation of mass, Navier stokes equation, Reynolds Averaged Navier Stokes equation (RANS) and additional turbulence modeling equations are also used in this analysis with appropriate boundary conditions. Hence flow is turbulent the different turbulence models are used in this analysis like standard-k-omega, k-omega-SST and transition-SST. CFD uses some numerical technique like finite element method (FEM), finite difference method (FDM), finite volume method (FVM) to solve the problem and get required solution.

Here in the present study using ANSSYS FLUENT -14 software ,with 2D -axisymmetric along with proper boundary condition and mesh convergence study used to get the require solution. In this study flow through pipe for turbulent flow is chosen for validation .Figure 2 shows geometry of flow domain is used in pipe flow for validation .Here axisymmetric model is used hence half of the portion is shown.

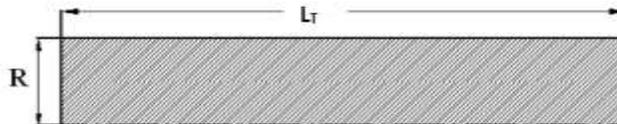


Fig 2 flow Domain Used in the Pipe Flow

Dimensions of pipe are follows

L_T = Total Length of the pipe = 4000 mm

R = Radius of the pipe = 20 mm

In the boundary conditions at the inlet of pipe flow velocity of 1 m/s and outlet of the pipe gauge pressure zero is defined. Here axisymmetric model is given hence pipe axis defined, water density is taken as 1000 kg/m³ and flow is considered as turbulent flow for Reynolds number 50000 and corresponding viscosity of fluid is calculated. Second order upwind method is used for analysis and convergence criteria 10⁻⁶ is given. Meshing of the geometry is done by using quadrilateral elements, number of mesh elements are 170000 and turbulence model transition-SST model has been selected.

From the standard equation for flow through pipe for turbulent flow, value of ratio of maximum velocity to the average velocity is 1.25 and the pressure drop in the pipe calculated from the Darcy – Weishback equation is 272.75 Pa over a 1m length of pipe line in the fully developed region. From the CFD results value of ratio of maximum velocity to the average velocity is 1.2456 and pressure drop (between 3m-4m) is 270.52Pa. Thus CFD results are in excellent agreement with the standard value and hence methodology used in CFD is validated.

PRELIMINARY DESIGN OF ROTAMETER

Now we will consider design of rotameter for metering flow rate of water in the range 10-100 lpm. Thus the input data for the design is as follows.

Design data

Density of water = $\rho_f = 1000 \text{ kg/m}^3$

Density of the steel = $\rho_s = 7900 \text{ kg/m}^3$

Maximum flow rate = $Q_{\max} = 100 \text{ lpm} = 1.667 \times 10^{-3} \text{ m}^3/\text{s}$

Viscosity of water = $\mu_w = 0.001 \text{ Pa-s}$

The schematic diagram of the Rotameter is given in Figure 3. The Nomenclature of various geometrical parameters are also given in the Figure.

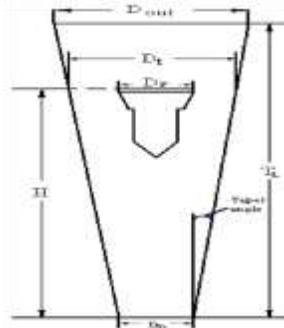


Fig 3. Geometrical Specifications of Rotameter

D_{in} = Inlet diameter of tapered tube

D_{out} = Outlet diameter of tapered tube

Θ = Half divergence angle of tapered tube

T_L = Vertical height of the tube

From the CFD analysis the value of coefficient of drag for a viscosity compensating float is taken as 2.4 and Reynolds number is taken as 50000.

The coefficient of drag = $C_D=2.4$ (assumed)

Reynolds number= $Re= 50000$ (assumed)

Consider the Reynolds number equation

$$Re = \frac{\rho v D}{\mu}$$

In the above equation the values of Reynolds number (Re), density (ρ), viscosity (μ) are known, but inlet velocity and inlet diameter of tube are unknown. In order to calculate the inlet diameter of tube the above equation can also be written as (maximum flow)

$$Re = \frac{\rho v D}{\mu} = \frac{4\rho Q}{\pi D\mu}$$

$$D = \frac{4 \cdot 1000 \cdot 1.667}{50000 \cdot \pi \cdot 1000 \cdot 0.001}$$

$$D = 42.44 \text{ mm}$$

From the above calculations inlet diameter of Rotameter tube is taken as approximately 40 mm. Since we know that when there is no flow in Rotameter, float is resting at the bottom of the tube therefore the maximum float diameter (D_f) is assumed as 40 mm.

Since inlet diameter of tube and flow rate fluid is known we can easily calculate the inlet velocity at the maximum flow by continuity equation

$$Q = (U_{in} \times A_{in})$$

$$U_{in} = 1.667 \cdot 10^{-3} / 1.256 \cdot 10^{-3}$$

$$U_{in} = 1.327 \text{ m/s}$$

Since velocity of fluid lies between 1-5 m/s, this is acceptable.

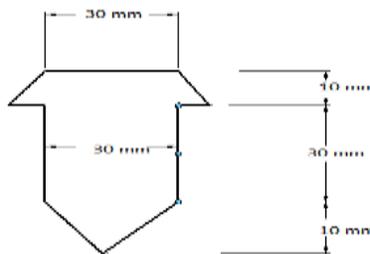


Fig 4. Float Dimensions in mm

Maximum diameter of float = $D_f=40$ mm

Inlet diameter of tapered tube = $D_{in}=40$ mm

Θ = Half divergence angle = $1-2^\circ$

Height of the float is assumed as = $h=50$ mm

Volume of the float $=V_s = \frac{\pi}{4} \cdot D_f^2 \cdot h = 62831.85 \text{ mm}^3$

Consider the equation (2.1) when the float is in equilibrium condition

$$F_D = W - F_B$$

Substitute corresponding equation (Eqn 4 and Eqn 5) for the above forces it become

$$C_D \cdot 0.5 \cdot \rho_f \cdot U_{anu}^2 \cdot A_s = (\rho_s - \rho_f) g V_s$$

$$U_{anu}^2 = \frac{6900 \cdot 9.81 \cdot 62831.85 \cdot 10^{-9}}{2.4 \cdot 0.5 \cdot 1000 \cdot (\frac{\pi}{4}) \cdot 0.04 \cdot 0.04}$$

$$U_{anu} = 1.6794 \text{ m/s}$$

From the continuity equation

$$Q = U_{in} \cdot A_{in} = U_{anu} \cdot A_{anu}$$

Where $e \quad A_{anu} = \frac{\pi}{4} (D_t^2 - D_{in}^2)$

$$Q = 1.6794 \cdot \frac{\pi}{4} (D_t^2 - D_{in}^2) \cdot \text{m}^3/\text{s}$$

From the trigonometric relation for the tapered tube

$$D_t = D_{in} + 2H \tan \Theta$$

Then equation becomes

$$Q = 1.6794 \cdot \frac{\pi}{4} [4D_{in}H \tan \Theta + 4H^2 \tan^2 \Theta] \cdot 60 \cdot 1000,$$

We know that $D_{in} = 0.04 \text{ m}$

$$Q = 79.139 \cdot 10^3 [0.16 \cdot H + 4 \cdot H^2 \tan^2 \Theta] \cdot \text{lpm}$$

In the above equation divergence angle is unknown, but usual practice its range from 1-2°. Below table shows Q and H value for two divergence angles of the tube.

Table 1 Position of the Float for Two Different Half Divergence Angles of the Tube

Q(lpm)	For $\Theta=1^\circ$	For $\Theta=1.5^\circ$
	H (mm)	H(mm)
10	44.27	28.64
20	86.9	57.29
40	168.57	112.68
60	245.71	164.24
80	319.20	213.90
100	388.57	259.74

In the normal design of Rotameter length of the tube is not to exceed around 350 mm. Hence $\Theta=1.5^\circ$ gives a proper height of tube. Thus $\Theta=1.5^\circ$ is taken for further design of Rotameter. However above relations are not exact since we have made simplifying assumptions during the design. The most important assumptions are that we assumed C_D to be independent of Re and d/D as well as it has a constant value of 2.4. Hence actual performance of Rotameter needs to

be evaluated either by calibration and / or computation by CFD. In the present study performance of Rotameter is analyzed using validated CFD methodology

Table II Position of the Float and Corresponding Inlet Velocity for Flow Rate of 10-100 lpm as Per Design Calculations($\theta=1.5$)

Q(lpm)	H(mm)	V _{in} (m/s)	D _T (mm)	D _F /D _T
10	28.647	0.1326	41.5	0.963
20	57.29	0.2652	43	0.919
40	112.24	0.5305	45.90	0.8714
60	164.24	0.7957	48.60	0.823
80	213.90	1.060	51.20	0.7812
100	259.74	1.3263	53.60	0.746

ANALYSIS OF FLOW THROUGH ROTAMETER USING CFD

The drag force calculated by using theoretical formula, when the float is in equilibrium position.

$$F_D = (\rho_s - \rho_f)g \cdot V_s$$

$$= (7900-1000) \cdot 9.81 \cdot 62831 \cdot 10^{-9}$$

$$= 4.253 \text{ N}$$

This is the drag force required to keep the float in equilibrium at any position inside the diverging tube.

A. Flow Domain, Geometry and Boundary Conditions of the Actual Rotameter for the CFD Analysis

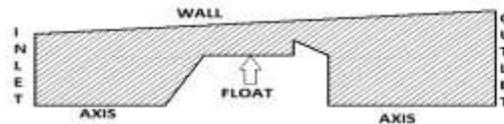


Fig 5 Geometry and Boundary Conditions of the Actual Rotameter

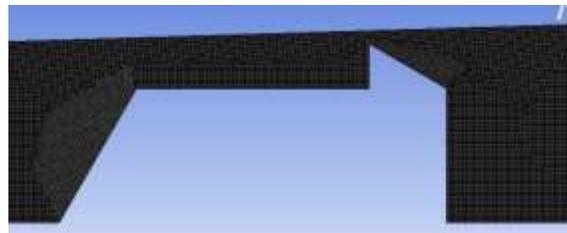


Fig 6 Fine Mesh around the Float Using Quadrilateral Elements

Fig 5 shows the geometry and boundary conditions of the actual Rotameter. At the inlet of the pipe velocity corresponding to flow rate is defined, at the outlet gauge pressure zero is defined and float and wall considered as stationary.

Fig 6 shows the fine mesh around the float using quadrilateral elements. Number of elements are around 170000 are used in this analysis

The float is placed in the position corresponding to the design flow rate, the inlet velocity corresponding to flow rate as per design calculation is given as input and drag force on the float is calculated. This need not be equal to the force required to keep the float in equilibrium, since the assumed value of C_D during design is only an approximate one. At the same position of float the inlet velocity is modified and computations are repeated until the drag force on the float becomes equal to 4.253 N. The flow rate corresponding to this inlet velocity is taken as the true flow rate

.This procedure is repeated for various positions of float corresponding to design flow rate range of 10-100 lpm in steps of 20 lpm .The model has been created in CFD as per design calculation and meshing of geometry is done by using quadrilateral elements and boundary conditions are also defined. Turbulence model standard k-omega model and simulation method second order upwind method is used and convergence criteria 10^{-6} has been given. Number of iterations for the convergence for was in the range 30000-40000 and time taken for each run was around 24 hours. The results obtained from CFD is tabulated below.

Table 3 Results of Flow Rate, Drag Force and Drag Coefficient Obtained from CFD

Q _{design} (lpm)	H (mm)	Q _{CFD} (lpm)	V _{new} (m/s)	F _D (N)	V _{anu} (m/s)	C _{D2}
10	28.647	9.9973	0.1323	4.247	1.471	2.62
20	57.29	20.109	0.2667	4.258	1.649	2.48
40	112.24	42.539	0.5642	4.248	1.837	2.43
60	164.24	67.41	0.8941	4.246	1.9489	1.779
80	213.90	94.099	1.2480	4.224	2.047	1.607
100	259.74	118.122	1.5669	4.234	2.247	1.60

Table 3 shows computed values of flow rate (Q_{CFD}) for various positions of float (H).As per the analysis the computed value of drag force equals the required value needed to keep the float is in equilibrium (4.253N),the computed value of coefficient of drag (C_{D2}) also given in the table. It is observed that actual value of cd is in the range of 1.6-2.48.Hence it is not constant as assumed in the design, due to the deviation between Q_{CFD} and Q_{design}. The deviation increases with increasing flow rate and at Q_{design} equal to 100 lpm actual flow rate is 118.1 lpm, deviation due to the assuming constant c_D for the float. It is seen from the Fig 7 also the deviation between design and actual value of increases with increasing in the flow rate.

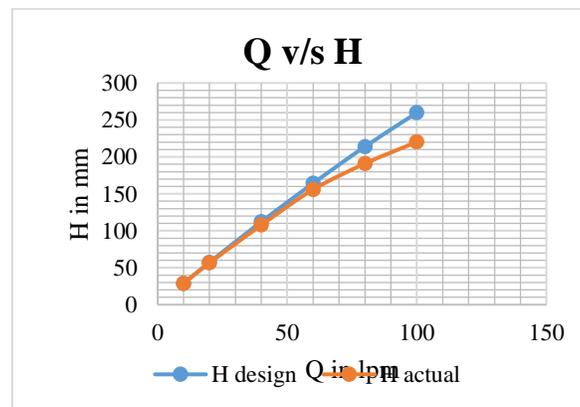


Fig 7 plot of position of float from design calculations v/s from CFD

Figure 8 shows the pressure contours for 80 lpm flow rate of water .We can observe that maximum pressure is developed in upstream region of the float and when fluid passes over the float reduction in the pressure take places along the length of tube .Because of less annulus area between the float and pipe sudden reduction in pressure takes place .

Figure 9 shows the velocity contours for 80lpm flow rate of water . it can be observed that flow separation take place around the float and maximum velocity is reached in the annulus region due to the small flow area.

Figure 10 shows the velocity vector plot for 80lpm flow rate of water, it can be observed that two vortices are formed beyond the float and they are counter rotating . Maximum velocity is reached in the annulus region due to the small flow area

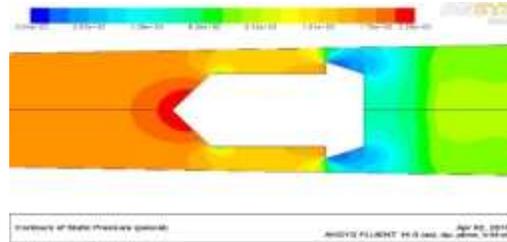


Fig 8 Pressure Contours for 80 lpm Flow Rate

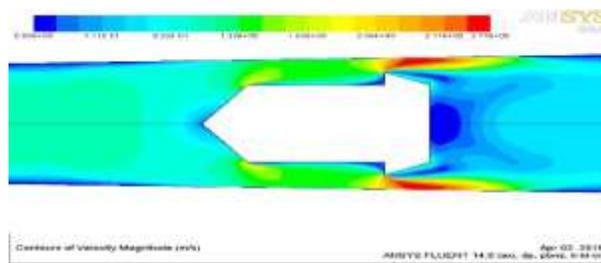


Fig 9 Velocity Contours for 80 lpm Flow Rate

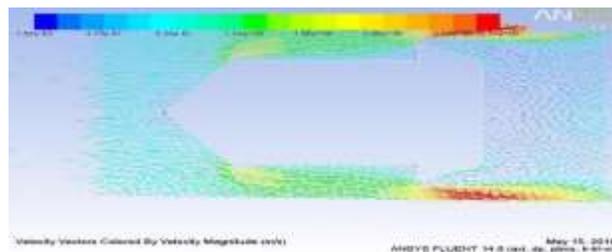


Fig 10 Velocity Vector for 80 lpm Flow Rate

B.Improved Design methodology of Rotameter

In the preliminary design of Rotameter the value of coefficient of drag is chosen by considering the effect of Re and d/D on the value of coefficient of drag as negligible and hence it is assumed to be constant. But after simulation is done in CFD, the C_D value obtained from each flow rate is different. Hence it shows that coefficient of drag is also dependent on the diameter ratio of float to the tube. Hence in this present study to evaluate coefficient of drag by considering effect of diameter ratio the following relation is proposed.

$$C_D = C_{D0} + \frac{K}{\left(1 - \frac{d}{D}\right)^n}$$

Where C_{D0}, K and n are constants

In order to calculate the values of k and n, an iterative process to find best fit values of these parameters. . For this purpose, the computed values of C_D at different positions as well as the corresponding d/D are used. By curve fitting, the best value of n is chosen as 0.3 and calculated value of K is 0.882. The value of C_{D0} is 0.4 and accuracy of the fit was +1% .the details of the calculations are available in the thesis of Deepu .The above equation is used to calculate the coefficient of drag for different diameter ratios corresponding to different flow rates and it is listed in below Table.

Table 4 Coefficient of Drag and Flow Rate Obtained from Equation

Q _{design}	H(mm)	d/D	C _{DCFD}	C _{eqn}	Q _{CFD}	Q _{eqn}
10	28.647	0.963	2.62	2.76	9.9973	9.017
20	57.29	0.911	2.48	2.21	20.109	20.52
40	112.24	0.871	2.43	2.02	42.539	43.85
60	164.24	0.823	1.779	1.87	67.41	68.27
80	213.90	0.781	1.607	1.78	94.099	93.84
100	259.74	0.746	1.64	1.72	118.122	118.98

Above Table 4 shows the results from the CFD. It can be observed that the value of C_D calculated based on calculation (C_{Deqn}) and C_D calculated from CFD analysis (C_{DCFD}) are in reasonable agreement. Using the values C_{Deqn} in the formula the flow rate corresponding to different positions of float are calculated and they are tabulated in the Table (Q_{eqn}). It is observed that agreement between Q_{eqn} and Q_{CFD} is within ± 1 lpm. Thus the revised design methodology using the correlation between C_D and d/D gives much more accurate design as compare to original methodology which assumed constant C_D

CFD ANALYSIS OF ROTAMETER PERFORMANCE UNDER NON-STANDARD CONDITIONS

A. Effect of upstream disturbance on the performance of Rotameter

During installation of Rotameter in a pipe line usually a valve is provided at the inlet, to isolate the valve when are it needed. Further some other pipe fitting like pipe bends sensors etc, these will disturb the flow at the inlet of the Rotameter and hence may effect the accuracy of Rotameter. In order to assess this effect CFD computations have been made.

In this study the effect of upstream disturbance on the performance of the Rotameter is modelled by placing a orifice plate inside the pipe nearer to the entrance of the tapered tube .The upstream disturbance distance is varied from twice of the inlet diameter of tapered tube ($2D_{in}$) to five times of the inlet diameter of tapered tube ($5D_{in}$). Here diameter of the orifice hole is taken as half of the inlet diameter of tube ($\beta=d/D=0.5$) and thickness of plate is taken as 3 mm. Boundary conditions are same as that of previous cases. Standard k-omega model and convergence criteria of 10^{-6} have been used.

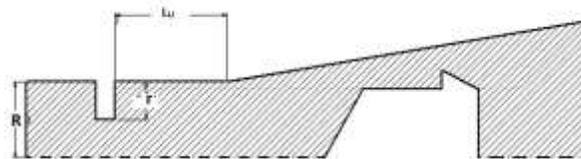


Fig 11 Upstream disturbance near the tapered tube β =diameter ratio =0.5

R- Radius of pipe =20 mm

r- Radius of plate=10 mm

L_U = Upstream length, 80 mm for $2 D_{in}$ and 200 mm for $5D_{in}$

Fig 11 shows the geometry of flow domain used to find upstream disturbance effect on performance of Rotameter .A orifice hole is placed near at the inlet of the tapered tube with a diameter ratio 0.5 boundary conditions are same as the previous cases of. The results obtained from CFD tabulated below.

Table 5 Effect of Upstream Disturbance on the Performance of the Rotameter At a Distance of $2D_{in}$

Q (lpm)	Q _{Disturbed} (lpm)	Q _{actual} (lpm)	Error (%)
10	9.9775	9.9973	-0.12
20	20.451	20.109	1.67
40	43.554	42.539	2.28
60	68.81	67.41	2.03
80	95.114	94.099	1.03
100	120.17	118.122	1.17

Above Table 5 shows that results obtained CFD for the upstream disturbance length of $2D_{in}$. The indicated flow rate for the disturbed condition ($Q_{Disturbed}$) is somewhat different from actual flow rate without disturbance (Q_{actual}). The variation in the in the range of 0.6-3.75%.

Table 6 Effect of Upstream Disturbance on the Performance of the Rotameter at a Distance of $5D_{in}$

Q (lpm)	Q _{Disturbed} (lpm)	Q _{actual} (lpm)	Error (%)
10	9.8559	9.9973	-1.23
20	20.3117	20.109	0.136
40	42.718	42.539	1.87
60	65.353	67.41	-2.77
80	94.71	94.099	0.65
100	122.73	118.122	3.75

Above Table 6 shows that results obtained CFD for the upstream disturbance length of $5D_{in}$. In this case error obtained in the range of 0.2-2.2%. Hence we consider maximum error 2% is acceptable, hence minimum disturbance distance required is $5D_{in}$.

B.Effect of Variation in Viscosity by Keeping Density Constant on the Performance of Rotameter

When the Rotameter is used for fluid other than the calibrated fluid, then correction factor for density variation is available in standard literature. However the correction factor for viscosity is not available, Pavan Kumar (1&2) has discussed detail regarding the adequacy of density correction factor. In the present study the effect of viscosity on the performance of Rotameter has been analyzed. In this study to find the effect of varying viscosity on performance of Rotameter computations are made by increasing the viscosity of fluid by 5 times and 10 times by while keeping density constant as 1000 kg/m^3 . Here analysis is done for flow rate range of 20-118 lpm . Positions of the float specified are same as those in the previous cases .Inlet velocity is calculated for corresponding flow rate of water and turbulence model standard k-omega has been chosen and convergence criteria 10^{-6} has been taken, the results obtained from CFD tabulated below.

Table 7 Performance of Rotameter for Liquid Of Viscosity 5 cP

Q _w (lpm)	F _D (N)	V _{in} (m/s)	Re	Q _{CFD} (lpm)	% variation
20.1	4.4443	0.2667	207.14	19.663	2.22
42.5	4.383	0.5642	4382.13	41.86	1.5
67.4	4.2482	0.8142	6944.46	67.43	0.04
94.1	4.1148	1.248	9693.98	95.66	1.62
118.1	3.882	1.5669	12170.09	123.61	4.65

From the CFD analysis computed flow rate (Q_{CFD}) and actual flow rate (Q_w) is not equal. This due to the fact that Re has changed, due to the change in viscosity of fluid. Revised calculations have been made for each position of float by changing the flow rate of fluid. The actual flow rate at which the drag force is equal to the equilibrium value (4.25 N) are calculated. the error measured in this case is less than 5%. Similar calculations are performed for a fluid with viscosity of 10 Cp and results are tabulated in below Table 9. It is observed that deviation in this case range of 0.6-3.72%. Thus it is necessary to derive expression for viscosity correction factor for this extensive study is required.

Table 8 Performance Of Rotameter for Liquid Of Viscosity 10 cP

Q_w (lpm)	F_D (N)	V_{in} (m/s)	Re	Q_{CFD} (lpm)	% variation
20.1	4.5253	0.2667	1035.72	19.48	3.18
42.5	4.4680	0.5642	2191.06	41.46	2.49
67.4	4.3071	0.8142	3472.2	66.97	0.64
94.1	4.2010	1.248	4846.91	94.68	0.61
118.1	3.9464	1.5669	6085.4	122.49	3.72

C. Performance of Rotameter Using Air as a Fluid

Rotameter designed in the present investigation is for metering the flow rate of water in the range of 10-100lpm. An attempt has been to study the performance of this Rotameter, when it is used to measure flow of air at different pressures. Computations have been made for air at 4 different pressures of 1atm, 2atm, 5atm and 10atm. and the density of air at each pressure is calculated from perfect gas equation. At each pressure computations have been made namely 20.1-118.1lpm. Simulations have been made for flow of air at these two positions of float for each inlet pressure. For all the cases viscosity is taken as 1.8×10^{-5} Pa-s. The computed values of drag force on the float are tabulated in the table. For each inlet pressure volumetric flow rate of air for these settings are calculated using density correction factor (Eqn2.16). The results obtained from CFD tabulated below.

Table 10. Performance Of Rotameter Using Air Flow

Atmospheric pressure	Density correction factor (c_s)	Q_w (lpm)	Q_{air} (lpm)	V_{in} (m/s)	F_D (N)	Q_{CFD} (lpm)	% variation
1	30.6	20.1	615.06	8.121	4.2495	614.80	0.04
		118.1	3613.86	47.93	4.848	3384.8	6.338
2	21.63	20.1	434.76	5.7404	4.2495	434.58	0.0414
		118.1	2554.5	33.87	4.8131	2401.2	6.008
5	13.679	20.1	274.95	3.6303	4.2456	274.71	0.087
		118.1	1615.48	21.42	4.753	1528.07	5.41
10	9.669	20.1	194.34	2.566	4.2443	194.14	0.10
		118.1	1141.90	15.144	4.559	1102.91	3.41

From the CFD results it is observed that computed value of drag force (F_D) are not exactly equal to required value (4.253N) for the equilibrium. This shows that density correction factor is not very accurate, the actual flow rate of air which gives the required drag force on float are listed in the table (Q_{CFD}) and deviation in the range of 1-6.34%.

If the Rotameter calibrated for water also used for air necessity application of density correction factor gives good results and error in the range of 1-6.34%.

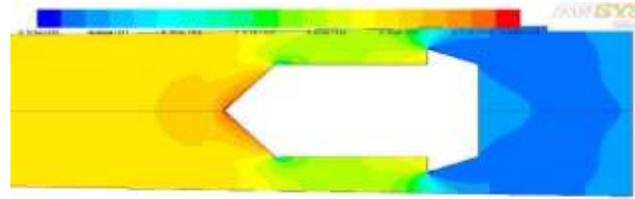


Fig 12 Pressure Contours for Air Flow at 1 atm

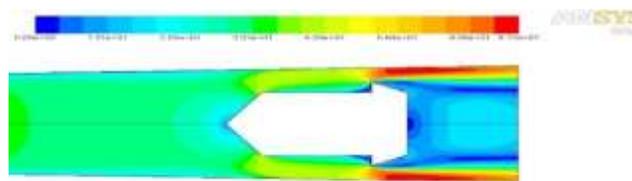


Fig 13 Velocity Contours for Air Flow at 1 atm

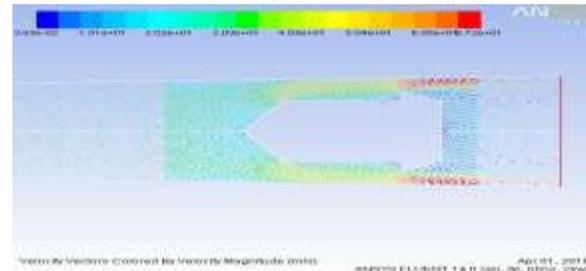


Fig 14 Velocity Vectors for Air Flow at 1 atm

CONCLUSION

1. ANSYS FLUENT 14 CFD software can be used for design and analysis of flow through Rotameter. However the Methodology has to be validated and suitable turbulence model and convergence criteria as well as discretization to be selected.
2. Design of a Rotameter based on simplified procedure by assuming coefficient of drag is constant has been made. The validating of this design has been verified by using CFD analysis.
3. Improved methodology for the design of Rotameter, by considering effect of diameter ratio (d/D) on the coefficient of drag has been proposed and validated.
4. CFD has been used to predict the performance of Rotameter with upstream disturbance and different liquids and gases. Thus CFD can be used to generate calibration data for existing Rotameter, when it is used for metering different types of fluids.

REFERENCES

- [1] Pavankumar M. tech thesis of "Prediction and Performance of Rotameter Using CFD.
- [2] Pavankumar K P, Dr V Seshadri, Puneethkumar 'Analysis of Flow Through Rotameter Using CFD', International Journal of Emerging Technology and Advance Engineering Volume 5, Issue 6, June 2015.

- [3] Rakesh Joshi, S.S.Jolly, BikramJit Singh “Effect of Float Design on the Performance of Rotameter using Computational Fluid Mechanics”, International Journal of Engineering and Management research, Volume-3, Page number: 15-20 Issue-6, December-2013.
- [4] Karna S Patel, Saumil B. Patel, Utsav B. Patel, Prof Ankit P. Ahuja, “CFD analysis of an Aerofoil International Journal of Engineering Research Vol-3.Issue.pp: 154-158,01 March 2014.
- [5] Sapna P.P Abdul Hakeem M. A “Drag Force Based Flow meter Design for Conducting Fluids”, International Journal of Research in Engineering and Technology, vol 3 issue 10, October-2014.
- [6] Kunal Chakraborty, N.Mandal, R,Sarkar “Design of an Electronic Flow Transmitter Using LVDT & Hall Sensor “by, International Journal of Electronics and Communication Technology Vol.4, Jan-March 2013.
- [7] D. Ramasamy, K. Kadirgama, A. K. Amiruddin and M. Y. Taib. “A Vehicle Body Drag Analysis Using by CFD”
- [8] Bogdan Stoyanov, Jordan Beyazov. “Determination of The Flow Rate of Different Fluids by a Rotameter”
- [9] Y Huang and J.Feng “Wall Effect on The Flow of Viscoelastic Fluid Around a Circular Cylinder” P., Journal of Non-Newtonian Fluid Mechanics, Pp: 179 – 198,1995
- [10] Kirmac and M Pakdemirli, “Analysis and Determination of Drag Forces for Spikes of Crops” Department of Mechanical Engineering, Mathematical and Computational Applications, Vol. 17, issue 1, pp: 83-91, 2012