Determining Flow Coefficient for Globe Valve with Different Trim shapes using a CFD Tool

Sahil Mahajan,
Pune Vidyarthi Griha's College of
Engineering and Technology,
Pune, Maharashtra – India

Dr. Naresh Jaiswal
Professor,
Pune Vidyarthi Griha's College of
Engineering and Technology,
Pune, Maharashtra – India

Abstract: **Globe valves are one of the most popular industrial valves used in process control applications. The main reason for their use in finite control element is due to their good throttling characteristics and low seat leakage with relatively high pressure drop thereby increasing the overall efficiency of the process. Hence it is very essential to study the inherent flow characteristics of the valve and to determine the best trim shapes for each application purpose. Using a Computational Fluid Dynamics code it is easier to understand such complex flows inside the valve. In this work a standardized globe valve is modeled and a CFD analysis of valve is done for three different trim shapes. Additionally, keeping the pressure drop and inlet flow-area constant and noting the change in other flow parameters for different valve opening positions the flow capability of the valve was found in each case.**

Keywords: Globe Valves, CFD, Valve Flow Coefficient.

I. INTRODUCTION

The control valves are mechanically operated devices that control flow parameters of the fluid flowing through the pipes. There are numerous industrial valves like Gate, globe, butterfly, ball, disc check valves, etc. each having their own specific characteristics and advantages over others for different flow applications. The process industries where frequent controlling of fluid flow, pressure, temperature and level is required make use of the globe valves owing to various advantages it possess over others. Globe valves show good throttling characteristics with very minimal seat leakage even for a very high pressure drop. Shorter stroke length (in comparison with gate valves) and easy assembly of parts are some other advantages which gives globe valves an edge over the others for flow control applications. The increasing demand for automated control valves in process industries has made a significant improvement in the designs of globe valves. For better flow characteristics and optimum valve sizing manufacturers have come up with changes in trim shapes [1]. The CFD based analysis of a ball valve in presence of cavitation was done to find out the changes in flow pattern at different valve opening angles. As opening of valve decreases a higher pressure drop is created which signifies the role of valve opening in determining the flow characteristics [2]. A numerical simulation of a globe valve with a pilot control is carried out to study the behavior of flow pattern for different conditions like closed, fluctuating, rapid opening of the valves and also with varying spring stiffness and inlet pressures in [3]. Another numerical analysis of three dimensional incompressible turbulent flows through high pressure drop control valves was carried out by using a CFD-ACE code to develop anti-cavitation control valve used in LNG marine system in [4]. An automatic mechanically operated safety valve was designed to cut water wastage and a CFD analysis in ANSYS Fluent solver was done to determine the flow parameters like pressure and velocity distribution for different mass flow rates which was later verified with the experimental results in [5]. A CFD analysis of a needle valve for different valve openings was done to carry out the flow coefficient calculations in [6]. A 3-D numerical simulation of stop valves was conducted to observe the flow pattern and the velocity-pressure distribution inside the valve along with additional investigation on wake induced vibration of valve and pipe system in [7]. A 2-inch globe valve was checked for five different cage models and flow coefficient was noted for each of them at full opening condition. The CFD and experimental results were found to be in good agreement and the cage with triangle shaped aperture proved to have the maximum flow coefficient value [8].

II. GLOBE CONTROL VALVE DESIGN AND SELECTION

The most common globe valve uses a T-style body, which allows the valve to be installed in a straight pipe with the top-works or actuator perpendicular to the line and will be used to explain the basic operation of a globe valve. Flow enters through the inlet port to the center of the valve where the trim is located. At this point, the flow must make a 90° turn to flow through the seat, followed by another 90° turn before exiting the valve through the outlet port [9]. The flow direction is often specified by the manufacturer and it is important to install the valve in the correct position on the pipe circuit. The main parts of the globe valve assembly are the valve body, trim, stem, cage and seat. New automatic control valves also have Pneumatic actuator, positioner with flapper nozzle assembly and other advanced features for accurate valve opening conditions required for processes in industries. The relationship between the valve opening and the flow coefficient is important for the selection of valves. Different trim shapes exhibit different flow characteristics and hence it is required to study each of them to find the best for a particular application.

One of the most important factors while valve selection and sizing is Flow coefficient or CV factor of the valve. It is defined as the number of US gallons of water per minute at 60F that will flow through a valve with a pressure drop of one psi. Its value helps to quickly compare different valve's flow

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 10, October-2018

capacities which in turn will help to select a properly sized valve for a desired application. Using the relations available it is possible to determine the pressure drop across the valve which is another important parameter during valve selection and sizing.

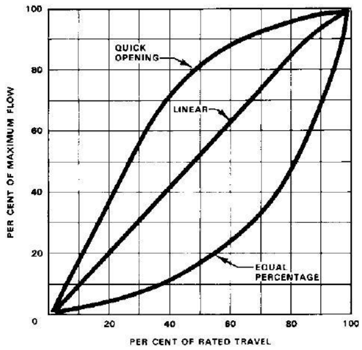

Fig. 1 Valve characteristics curves

Trims are generally categorized into three types viz. quick opening, linear and equal percentage. The valve characteristics from valve handbook for three different trim shapes are shown in figure 1. Mathematically the flow coefficient can be expressed as

$$C_v = Q * \sqrt{(\frac{SG}{\Delta P})}$$

Where, Q = discharge in gpm

SG = Specific gravity of the fluid

ΔP = Pressure drop

III. PROBLEM DEFINITION AND OBJECTIVE

In this project, a work is presented on a 125NB two-way globe valve to calculate flow coefficient for different trim types. Basically, three trim shapes showing on-off, linear and equal percentage characteristics of valves were modeled in Solidworks and later used to do CFD analysis in Ansys Fluent using each of them. The velocity and discharge were measured at an interval of 10mm lift for a constant pressure drop of 5 bar. The inlet and outlet gauge pressure considered was 10 bar and 5 bar respectively throughout all the iterations. The main objective of this work was to analyze the flow characteristics for different trim shapes using a CFD tool and then plot a graph of Discharge vs. Valve opening. Moreover, it will provide the idea of trim selection for various flow conditions that are required in industry.

IV. MODELLING AND SIMULATION

Solid Modeling

A 125 NB class 300 globe valve along with different internal parts and mainly the three different types of trim shapes were modeled in Solid works. Individual assemblies were done for each and for simplicity of flow analysis the stem and bush of each plug was suppressed to avoid unnecessary complexity during flow analysis.

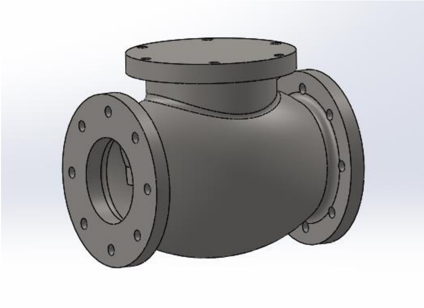

Fig 2: Globe Valve body

V. COMPUTATIONAL FLUID DYNAMICS

For doing the flow analysis of this model, there is a requirement of a CFD tool. It uses numerical methods and algorithms to solve and analyze problems with fluid flow using the governing equations and the boundary conditions specified. For this work ANSYS Fluent was used to model the problem.

Methodology

The assembly file from CAD software was converted into IGES and imported into design modeler. The fluid domain created, took the shape of the entire void. The meshed volume of fluid domain is as shown in Figure 3. The gap between the trim surface and valve seat had a very fine mesh which was necessary to capture accurately the flow through that narrow region. The mesh statistics are given in the following table.

Domain	Nodes	Elements
Valve body	68526	36113
Fluid domain	28315	147015
Trim	13555	69540
Total	110396	552668

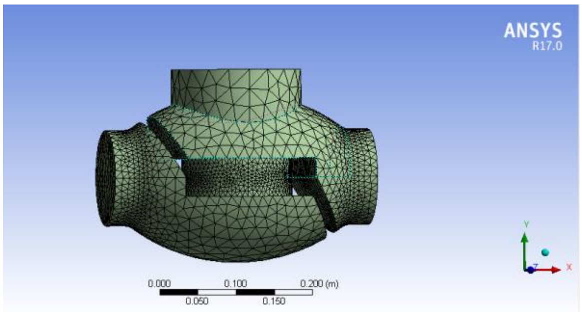

Fig 3: Fluid domain volume mesh

The CFD code used in Fluent is based on Finite Volume technique wherein a complex geometry can be divided into finite regions of control volume and the mass and momentum conservation equations are applied to each control volume. The governing equations for steady state flow condition of incompressible fluid are as follows :

Continuity equation :

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 10, October-2018

Momentum conservation equations:

$$\rho\frac{Du}{Dt} = \rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

$$\rho\frac{Dv}{Dt} = \rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

$$\rho\frac{Dw}{Dt} = \rho g_w - \frac{\partial p}{\partial w} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right)$$

In addition to the above Navier- Stokes equations, transport equations for turbulent kinetic energy (k) and the rate of dissipation of energy (e) were used to solve the problem. The Realizable k-epsilon model accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, strong adverse pressure gradients, separation and recirculation. Further, the default Standard wall treatment as recommended for industrial flows was selected [10].

Transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_m + S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_j}(\rho\epsilon\mu_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \epsilon}{\partial x_j}\right] + \rho C_1 S\epsilon - \rho C_2(\frac{\epsilon^2}{k + \sqrt{\vartheta\epsilon}})$$

$$+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}G_b + S_\epsilon$$

Where, the model constants are $C_2 = 1.9,\ \sigma_k = 1.0,\ \sigma_\epsilon = 1.2\ and\ C_{1\epsilon} = 1.44,$

The fluid that was used in this problem is liquid water whose properties were acquired from Fluent database. The boundary conditions of pressure inlet of 10 bar (gauge) and pressure outlet 5bar (gauge) were given input which gave the required pressure drop of 5 bar as needed for the case. Appropriate mesh interfaces were selected for different contact regions. Initially it generated a lot of errors but after simplifying the geometry and avoiding unnecessary components the problem was resolved. Hence in the final analysis, the results were obtained with the seat incorporated as the part of valve body. Also the stem of the plug was omitted. Incorporating these changes made sure that the solution converged for a much lesser number of iterations thereby reducing computing time with no significant effect on the results.

Finally, the pressure velocity coupling method used was SIMPLE which stands for Semi-Implicit Method for Pressure Linked Equations. It is widely used to solve the Navier-Stokes equations and extensively used by many researchers to solve different kinds of fluid flow and heat transfer problems. For higher accuracy, second order upwind scheme of spatial discretization was selected for pressure, momentum and k and epsilon equations at cell centroid. The Residual monitors for equations were set for convergence criteria of 1*e-4.

VI. SIMULATION RESULTS

The first analysis was done for the lift of 20% valve opening and later was solved for subsequent increase of 10mm upto the maximum lift. Using area weighted average the velocity magnitude at outlet for each interval was recorded. Furthermore, the mass flow rate obtained was used to calculate the KV and CV factor for each iteration for all the trim shapes.

Results for trim shape 1

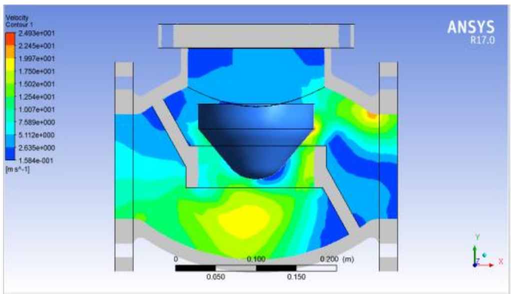

Fig 4: Velocity contour for valve opening of trim 1 at 40%

Lift (%)	Outlet Velocity (m/s)	Discharge(m3/hr)	Kv	Cv
20	6	264.81	118.42	136.90
40	8.2	361.91	161.85	187.10
60	9.46	417.52	186.72	215.85
80	10.25	452.39	202.31	233.87
100	10.5	463.42	207.25	239.85

Table 1: Result table for trim shape 1

Results for trim shape 2

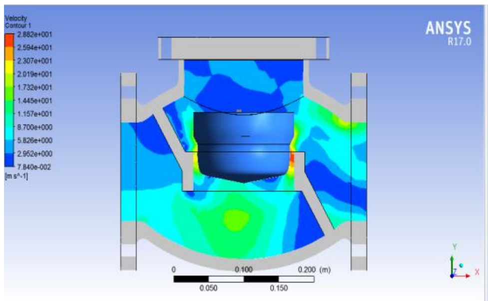

Fig 5: Velocity contour for valve opening for trim 2 at 40%

Lift (%)	Outlet Velocity (m/s)	Discharge(m3/hr)	Kv	Cv
20	5.18	225.09	100.66	116.36
40	5.6	247.16	110.53	127.77
60	6.1	269.22	120.40	139.18
80	6.7	295.71	132.24	152.87
100	8.61	380.01	169.94	196.45

Table 2: Result table for trim shape 2

Results for trim shape 3

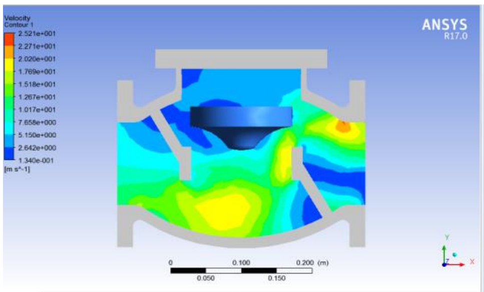

Fig 7: Velocity contour for valve opening of trim3 at 40%

Lift (%)	Outlet Velocity (m/s)	Discharge(m3/hr)	Kv	Cv
20	7.21	318.22	142.31	164.51
40	8.26	380.01	169.94	196.45
60	9.36	413.11	184.74	213.57
80	9.93	438.27	196.00	226.57
100	9.96	439.59	196.59	227.26

Table 3: Result table for trim shape 3

VII. COMPARISON OF RESULTS

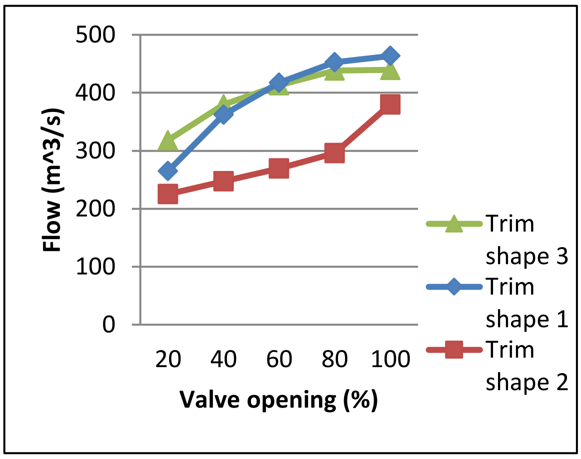

Fig 8: Graph of flow Vs percent valve opening

For trim shape 1, the rate of increase in flow rate is directly proportional to the opening of the valve. Hence it can be used where linear valve characteristics are required. A typical application for such characteristics can be process where constant pressure drop is expected for different flow rates.

The curve for trim shape 3 shows an initial rapid increase in the flow for less percentage of valve openings. After 50% valve opening there is very slight change in the flow with increase in lift. Thus valves with this type of trims installed tend to show a fast opening characteristic. They find their use in On-off control applications.

Lift (%)	Change(%)	Discharge(m3/hr)	Change(%)
20		225.096	
	20		9.80
40		247.1616	
	20		8.92
60		269.2296	
	20		9.83
80		295.7112	

Table 4: Equal percentage change in flow rate

From table 4 it can be inferred that trim shape 2 shows an equal percentage characteristics. For each 20% change in the opening of the valve there is approximately 10% of the change in flow rate. Additionally, from graph it is noted that valves with parabolic trim shapes are slow opening valves i.e. they show a very less change in flow rates for initial increase in lift. Owing to this fact such trims can be used for low flow rate applications and where control output stability is of prime importance.

VIII. CONCLUSION

In this work, flow coefficient, CV, one of the important factor for valve sizing was determined for different trim shapes using a commercial CFD code. The flow coefficient and discharge was calculated for each trim shape for different valve opening stages which on comparison showed the classic trends of flow pattern for linear, fast opening and equal percentage characteristics of a globe valve. Furthermore, it also showed that trim shape with equal percentage characteristics have tendency of low flow rates at the start and overall has less discharge capacity for same lift as compared to valve with linear characteristics. This work also shows that CFD techniques can be implied for checking various valve flow characteristics performing a lot of virtual iterations thereby providing improvement in the accurate designing of valve trims for specific requirements.

REFERENCES

[1] Emerson Process Management Control Valve Handbook.
[2] A. S. Tabrizi, M. Asadi , G. Xie , G. Lorenzini, and C. Biserni. Computational Fluid-Dynamics-Based Analysis of a Ball Valve Performance in the Presence of Cavitation ISSN 1810-2328, Journal of Engineering Thermophysics, 2014, Vol. 23, No. 1, pp. 27–38. c Pleiades Publishing, Ltd., 2014.
[3] Jin-yuan Qian, Lin Wei, Zhi-jiang Jin, Jian-kai Wang, Han Zhang, An-le Lu. CFD analysis of the pilot-control globe valve. Energy Conversion and Management 87 (2014) 220-226.
[4] Young JoonAn, Byeong Jin Kim and Byeong Rog Shin. Numerical analysis of 3-D flow through LNG marine control valves for their advanced design. Journal of Mechanical Science and Technology 22 (2008) 1998~2005
[5] Muhammed Safa Kamer, Ahmet Kaya, Abdullah Sisman. Experimental And Numerical Investigation Of The Flow Analysis Of The Water-Saving Safety Valve. INTERNATIONAL JOURNAL OF SCIENTIFIC & TECHNOLOGY RESEARCH VOLUME 4, ISSUE 10, OCTOBER 2015 ISSN 2277-8616
[6] Romil Khowal, D N Jadhav, Tansen Chaudhari. Flow Coefficient Valve Calculation using CFD Analysis for Needle Valve. International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME) V-5 No.2 ISSN (Print): 2319-3182
[7] Qin Yang, Zhiguo Zhang, Mingyue Liu, Jing Hu. Numerical Simulation of Fluid Flow inside the Valve. Procedia Engineering 23 (2011) 543 – 550.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 10, October-2018

[8] Sreekala S. K., Thirumalini S.Study of Flow Performance of a Globe Valve and Design Optimisation.Journal of Engineering Science and Technology Vol. 12, No. 9 (2017) 2403 - 2409 © School of Engineering, Taylor's University

[9] Valve Handbook by Skousen,Valtek International. McGraw Hill,New York San Francisco Washington. D.C. Auckland .Bogota Caracas Lisbon London Madrid Mexico City Milan Montreal New Delhi San Juan Singapore Sydney Tokyo Toronto, 729 p.

[10] Ansys Fluent Tutorial guide. Release 17.0. Ansys Inc.

[11] www.cfd-online.com. CFD-online discussion forum.

[12] Forbes Marshall Control Valve Brochure. Opp 106th Milestone Bombay Poona Road, Kasarwadi, Pune - 411 034. INDIA.

[13] Adam Del Toro. Master's thesis on COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF BUTTERFLY VALVE PERFORMANCE FACTORS. Utah State University. Logan, Utah. 2012.

[14] Martin Turesson. Master of Science thesis on 'Dynamic simulation of check valve using CFD and evaluation of check valve model in RELAP5'.Department of Chemistry and Bioscience Chalmers University of Technology SE-412 96 Göteborg, Sweden.

[15] Control Valve Sizing Coefficients. Published by METSO Flow Control Inc. P.O. Box 304, FI-01301 VANTAA Finland.

[16] Vb1 Engineering Data book. Johnson Controls.507 E. Michigan Street P.O. Box 423 Milwaukee, WI 53201.