C_v Value Evaluation of a Globe valve using CFD Analysis

S. K. Sreekala*, U. Muthukumar*

* Fluid Control Research Institute, Kanjikode west, Palakkad, Kerala, India.

Email: s.k.sreekala@fcriindia.com

Abstract

A globe valve is a linear motion valve used to stop, start and regulate fluid flow. Caged type globe valves are used for many industrial applications due to its better throttling capability and lesser leakage as compared to other types. In cage type Globe valves, cage design and configuration have a great impact on flow performance characteristics. The computational study of a cage type globe valve was carried out using FLUENT, a finite volume based code to find out the C_v value at ten openings and to validate with the experimental results. Grid sensitivity tests were done and the results were validated experimentally. Computational results were found to be in good agreement with the experimental results. Hence flow characteristics were studied using computational analysis.

Key words : *flow coefficient, globe valve, caged, throttling*

1. Introduction

Cage type globe valve is most commonly used control valve for high pressure system due to better throttling characteristics, lesser leakage and higher erosion resistance as compared to other valves. In cage type Globe valves, cage design and configuration have a great impact on flow performance characteristics. Hence in order to achieve a better flow performance, a detailed investigation of design and configuration of these types of valves are imperative. Globe valves are also recommended for services requiring frequent operation and positive shutoff. Chern and Wang [1] investigated the effects of V-port on the volume flow rate and flow features. 3-D numerical simulations and experiments were conducted to observe the flow patterns and to measure performance coefficients when V-ports with various angles were used in a piping system.

Huang and Kim [2] simulated turbulent flows in a butterfly valve using FLUENT, in which the k-ε model was employed for turbulence modeling. Merati, et al. [3], investigated the flow around a V-sector ball valve. Flow coefficient determination is an important factor in design of valves. Yogesh Gawas et al. [4] performed numerical simulations using commercial code FLUENT, to study and to estimate the valve sizing coefficient, torque coefficient and cavitation index for investigation of cavitating. Sreekala and Thirumalini [6] have carried out noise evaluation and experimental validation in globe valves using FLUENT for flow performance study. In the present investigation, Globe valve having cage configuration with circular apertures and different throttle positions are modeled to find out the valve coefficient, pressure and velocity contours inside and outside the cage and is validated with experimental results.

2. Computational Analysis

Computational Fluid Dynamics (CFD) is extensively used for research and development as well as design-oriented tasks in industry. CFD is an important tool in the design and development of new products. CFD is increasingly more cost effective than wind tunnel testing and it requires a lower energy consumption.

Flow characteristics study and design optimization of globe valves is a growing area of research. Incompressible flow simulation was carried out using FLUENT to find out the valve coefficient and to study the flow characteristics inside the valve. Computational Fluid Dynamics (CFD) analysis was carried out for a 2" Globe valve having cage with eight circular apertures arranged in two rows in zig zag manner. Flow medium considered is water at standard conditions and 3D steady incompressible flow solver of FLUENT is used for flow simulation. FLUENT is a finite volume based software package and is suitable for analysis of all types of fluid flows. In the present study all geometries are modeled using GAMBIT, which is the pre processor of FLUENT. Cages with eight circular apertures are modeled and simulated for various differential pressures and flow rates and at different opening conditions. Figure 1 shows the valve with cage having circular apertures at full opening condition.

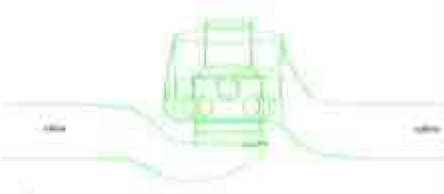

Figure 1. Valve domain

Computational domain of the valve is extended up to 10D (D is diameter of pipe) upstream and 15D downstream of the valve to get developed flow at both ends. Pressure inlet and pressure outlet boundary conditions are used for the inlet and the out let of the valve respectively for simulation. No slip boundary conditions are applied at the walls. Grid independent study was carried out with mesh size varying from 1.5 millions, 2.5 millions, 2.9 millions and 3.2 millions for which C_v values varied from 62, 60, 58.5, 58.2 respectively. C_v values were found to be varied within 1%, for valve mesh size of 3.2 millions as compared with mesh size of 2.9 millions and hence mesh size of 2.9 millions was considered for simulation. Figure 2 shows the mesh model of the valve.

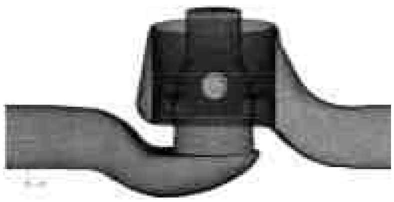

Figure 2. Mesh

RNG k-ε turbulence model is used for turbulence modeling. RNG Model follows the two equation model and is derived from the fundamental governing equations for fluid flows using Renormalization Group (RNG) theory. Flow through the valve is governed by three dimensional Navier Stoke's equations, continuity and three momentum equations. Segregated solver of FLUENT is used for simulation. Iterations were carried out till all the residuals are less than 0.00001.

Valve at different opening conditions under differential pressures were considered for simulation and valve coefficient (C_v) values are evaluated for comparison. The area of the flow passage is fixed for all cases as 0.000346185 m^2. Figure 3 indicates variation of C_v with percentage of stroke at ten openings.

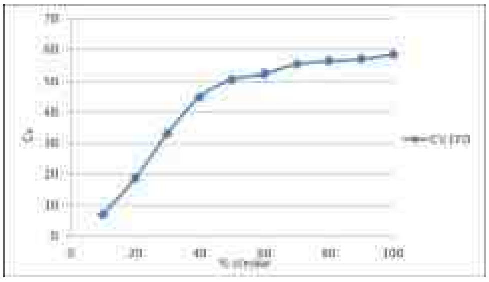

Figure 3. % Stroke vs C_v

3. Experimentation

The valve was installed in the test line as indicated in Figure 4 ,which shows the Schematic of test setup.

Figure 4. Experimental setup

The valve was set at the fully open position. The downstream control valve was set to give the desired pressure differential across the test valve, which was measured using high precision differential pressure transmitters. When flow conditions had stabilized, the flow rate was measured using the reference flow meter and differential pressure across the valve was also noted. Upstream pressure(P_{up})and temperature were also recorded. The following standards were referred

ISO 4185-1980: “Measurements of Liquid flow in closed conduits using weighing method”

ANSI/ISA 75.02-1996 : “Control Valve Capacity Test Procedure”

ANSI/ISA 75.01.01-2002 : “Flow Equations for Sizing Control valves

At each opening, flow rate was measured at three differential pressures, 0.9bar, 0.5bar & 0.1bar and the procedure was repeated for each valve opening. Test was conducted at openings varying from 10% to 100% at an interval of 10%. At each opening, valve coefficient(C_v) was found for various differential pressures using the equation(1).

$$C_v = \left[\frac{Q}{0.865}\right] * \sqrt{G/D_p} \qquad (1)$$

Where Q is the flow rate (m^3/hr) ,G is specific gravity and D_p is differential pressure across the valve (bar). Temperature of water was noted during each test using on-line Resistance Temperature Detector (RTD) with sensitivity of 0.01^0 C. The line pressure was measured using a precision pressure gauge with precision of 0.01bar and density of water was obtained from an on-line Densitometer with 0.001 kg/m3 precision. Experimental and computational results are compared in Fig. 5. % Error with respect to experimental results were evaluated and was found to be less than 9% for all cases.

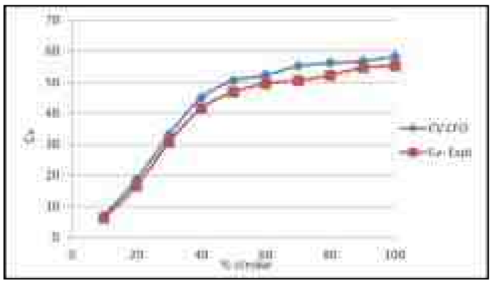

Figure 5. Validation of C_v values

4. Results and conclusions

Flow characteristics of the globe valve were predicted using computational analysis and validated with experimental results. The analysis was carried out for a 2” globe valve having cage with 8 circular apertures at ten openings,10%-100%. Computational results were found to be in good agreement with the experimental results and the percentage error was found to be less than 9%. The present work can be further extended with modifications in the area and cage aperture configurations for design optimisation.

References

[1] Chern, M. J., Wang, C. C., 2004. Control of Volumetric Flow-Rate of Ball Valve Using V-Port, Journal of Fluids Engineering **126**, 471-481.

[2] Huang, C., Kim, R. H., 1966. Three-Dimensional Analysis of Partially Open Butterfly Valve Flows, Journal of Fluids Engineering **118**, 562–568.

[3] Merati, P., Macelt, M. J., Erickson, R. B.,2001. Flow Investigation Around a V-Sector Ball Valve, Journal Fluids Engineering **123**, 662–671, DOI: 10.1115/1.1385831.

[4] Yogesh Gawas, Vilas R. Kalamkar and Vijay Mali, 2010. Modelling and Simulation of Valve Coefficients and Cavitation, Proceedings of the 13th Asian Congress of Fluid Mechanics 17-21 December 2010, Dhaka, Bangladesh, pp. 565-568.

[5] Yakhot,V., Orszag, S. A., Thangam, S. , Gatski, T. B., Speziale, C. G. ,1992.Development of turbulence models for shear flows by a double expansion technique, Physics of Fluids **4**, 1510–1520.

[6] Sreekala, S. K. and Thirumalini, S. ,2016,Investigation on Aerodynamic Noise Evaluation and Attenuation In a Globe Valve using CFD Analysis. International Journal of Multi physics, 10(1), 43-52

[7] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics- The Finite Volume Method, Prentice Hall, 1995

[8] J. D. Anderson, Computational Fluid Dynamics: The Basics with Applications, McGraw Hill, 1995

[9] Control Valve Capacity Test Procedures, ANSI/ISA–S75.02-1996