World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:16, No:11, 2022

A Study on Improving the Flow Capacity of the Valves

A. G. Pradeep, Gorantla Giridhar Kumar, Vijay Turaga, Vinod Srinivasa

***Abstract*—**The major problem in the flow control valve is of lower Flow Capacity (Cv) which will reduce overall efficiency of flow circuit. Designers are continuously working to improve the Cv of the valve, but they need to validate the design ideas they have regarding the improvement of Cv. Traditional method of prototype and testing take a lot of time, that is where CFD comes into picture with very quick and accurate validation along with the visualization which is not possible with traditional testing method. We have developed a method to predict Cv value using CFD analysis by iterating on various Boundary conditions, solver settings and by carrying out grid convergence studies to establish correlation between the CFD model and Test data. The present study investigates 3 different ideas put forward by the designers for improving the flow capacity of the valves like reducing the cage thickness, changing the port position, and using the parabolic plug to guide the flow. Using CFD, we analyzed all design changes using the established methodology that we developed. We were able to evaluate the effect of these design changes on the Valve Cv. We optimized the wetted surface of the valve further by suggesting the design modification to the lower part of the valve to make the flow more streamlined. We could find that changing cage thickness and port position has little impact on the valve Cv. Combination of optimized wetted surface and introduction of parabolic plug improved the Cv of the valve significantly.

***Keywords*—**Flow control valves, flow capacity, CFD simulations, design validation.

I. Introduction

CONTROL valves are devices that control flow parameters of a fluid flowing through the pipes. Valves have many applications in industry to control different types of fluids (water, steam, oil and gases). Most of the modern plants depend on correct distribution of flowing liquids or gases. Performance of control valves has direct impact on the stability and efficiency of entire system.

There are different types of valves depending on the application and the type of fluid used. Our focus will be on a type of valve called globe valve which is primarily designed to start/stop or regulate flow. It is comprised of a movable valve stem and a stationary seat in a generally spherical body. The working principle of globe valve is shown in Fig. 1.

Globe Valve Body Patterns

There are three primary body patterns or designs for Globe valves as shown in Fig. 2.

To describe the performance of a valve, flow coefficient (Cv) is commonly used. Higher the flow coefficient, higher is the energy efficiency. A flow coefficient primarily describes the relationship between pressure drop across the valve and the corresponding flow of fluid. Measurement of static pressure across valve for calculating CV is shown in Fig. 3.

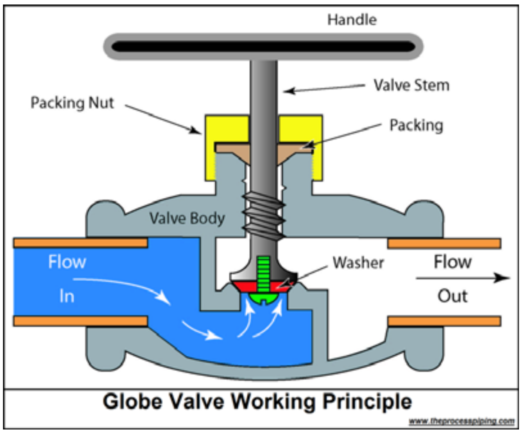

Fig. 1 Globe valve working principle [6]

Mathematically the flow coefficient Cv (or flow capacity rating of valve) can be expressed as:

$$Cv = Q\sqrt{S.G\Delta P}$$

where Q is the rate of flow (expressed in US gallons per minute); SG is the specific gravity of the fluid (for water = 1); ΔP is the pressure drop across the valve (expressed in psi).

Significance of Valve Flow Coefficient (Cv) Calculation

Calculating the Cv for a valve is very important as it will be easy to compare the different valves flow capacity by measuring control valve flow coefficient. It assists in selection of the right size control valve for a particular application. We can easily understand the valve characteristics by calculating Cv. Therefore, it is easy to determine the effect of valve on the pressure in the system.

A convenient method to predict Cv is required. Evaluation of flow coefficient can be done through theoretical calculation or experimental measurement. Theoretical calculation does not include the effect of valve shape and structure and hence not very accurate. Experimental measurement results in significant cost and time and, experimental method can only provide limited information and the interaction between valve structure and flow characteristics cannot be evaluated.

Pradeep Aadi Gopala Krishna is with HCL Technologies limited, India (e-mail: pradeep.ag@hcl.com).

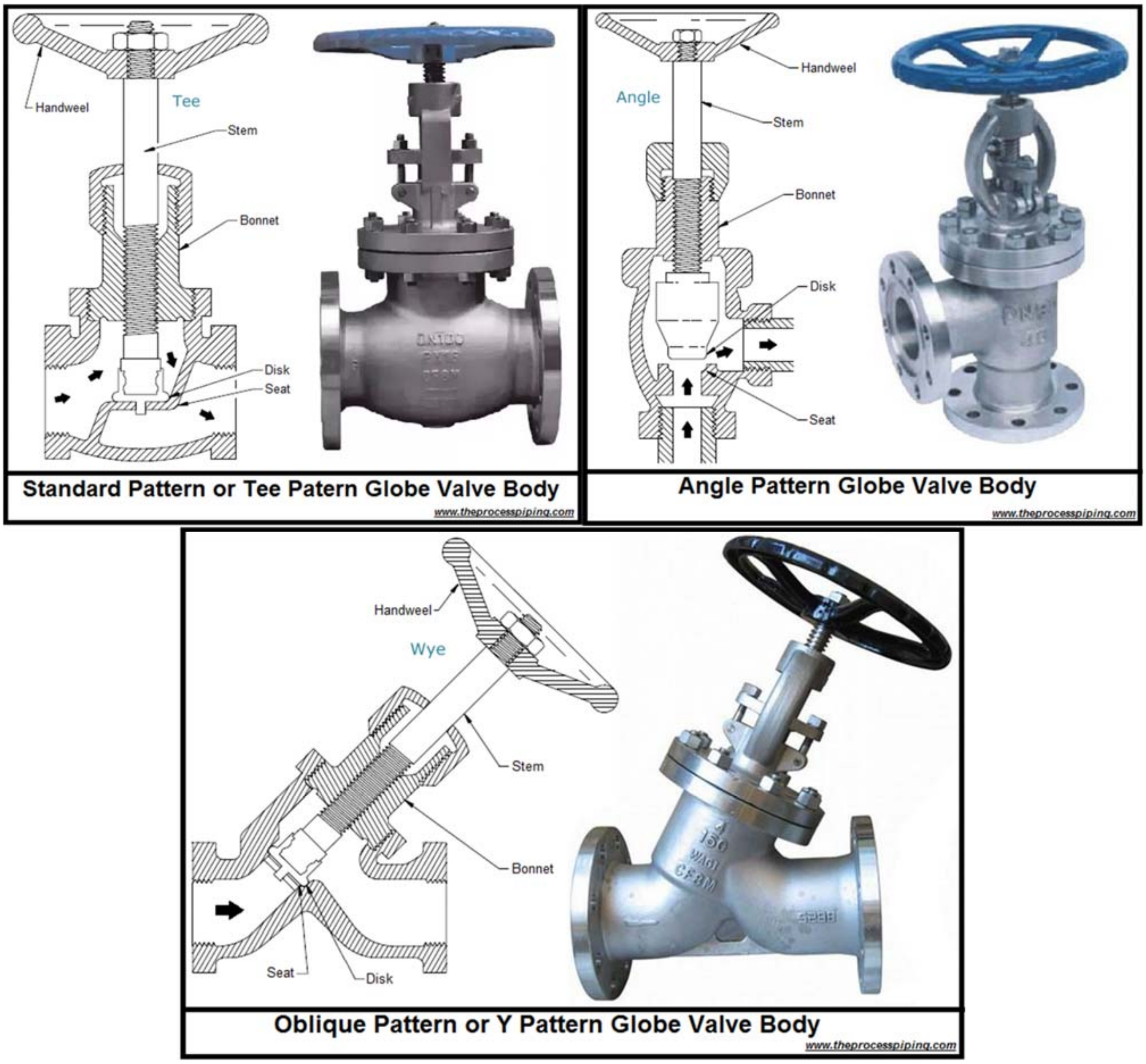

Fig. 2 Types of designs for Globe valve [6]

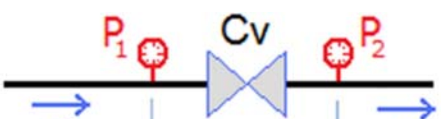

Fig. 3 Measurement of Static pressures across valve for calculating Cv [8]

Recently, Computational Fluid Dynamics (CFD) has been effectively used to predict the Cv as it helps in visualizing flow field and helps designers to explore various design options. Thus, CFD simulations are effective alternatives to experimental methods to evaluate Cv of valves. As we decided to use CFD for evaluating the ideas we had for improving the design of the valve, we looked for similar studies carried out using CFD. We found that [1] performed computational study of a cage type globe valve using FLUENT code to find out the Cv value at ten different openings and validated the results with the experimental results. Computational results were found to be in good agreement with the experimental results and the percentage deviation was less than 9%. Zhou et al. [2] proposed a new CFD method to calculate Cv using a linear fitting method to estimate the net pressure loss induced by the pipe and then that induced by the valve. Mahajan and Jaiswal [3] estimated Cv for different trim shapes using FLUENT code. The flow coefficient and discharge were calculated for each trim shape for different valve opening stages. Praveen and Pathan [4] presented 3-dimensional numerical simulations and experiment conducted to observe the flow pattern and to measure valve Cv when globe valve with different flow rate and uniform incoming velocities were used in a pipeline system. Yevale and

Gharge [5] presented modeling and simulation of the different control valves. We could understand how to evaluate Cv for the valves and general methods to improve it by referring to some articles online [6]-[8] as well. The results of this paper provide understanding of the flow pattern of the valve with different flow rate and determines the methods which could be adopted to improve the performance of the valve.

II. Methodology

We considered four design changes in 12-inch baseline globe valve design, modifying the lower cage (Iteration1), modified lower cage with increment in cage thickness (Iteration2), modified lower cage with change in port position (Iteration3), and modified lower cage with parabolic (Iteration4) plug shown in Fig. 4. The comparison study was made for all the five domains using CFD analysis.

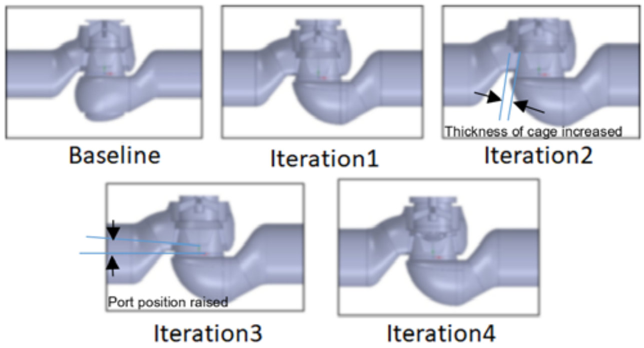

Fig. 4 Fluid domain used for CFD analysis

Inlet pressure of 5 bar and outlet pressure of 3 bar were applied. Water was considered as working fluid. Simulation was carried out in steady state and turbulence model of k-epsilon realizable with standard wall functions has been used.

Plane upstream at 2D and Plane downstream at 6D are used to calculate Cv as shown in Fig. 5.

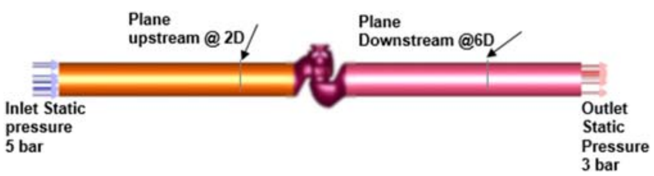

Fig. 5 Flow direction (flow to close)

III. Numerical Modeling

The fluid domain for the valve geometry was extracted and the domain on both inlet and outlet side were extended by 10 times the diameter of the pipe using ANSYS Space claim tool shown in Fig. 6.

The poly-hex core grid generation with boundary layer at fluid wall was done using ANSYS FLUENT. A view of the mesh is shown in Fig. 7.

Grid Independence Study

We carried out a grid independence study with cell count ranging from 5600 to 3.35 million. We chose element length 3 mm (Mesh10) considering tradeoff between accuracy and computational time and ensured that our conclusions and Cv evaluations are independent of the size of the grid we used for the simulations shown in Table I.

Fig. 6 CAD model

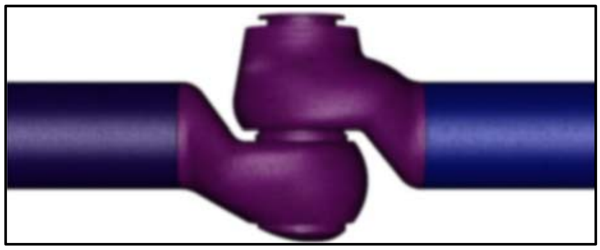

Fig. 7 Polyhexagonal grid

TABLE I
Grid Independence Study

Variable	Element length (mm)	Size	Massflow rate	Cv
Mesh01	50	5600	211.37	1240
Mesh02	40	8222	213.17	1250
Mesh03	30	16055	205.13	1200
Mesh04	20	40252	203.74	1195
Mesh05	10	211154	224.38	1301
Mesh06	8	360552	222.50	1231
Mesh07	6	783468	220.05	1240
Mesh08	5	1440579	218.14	1239
Mesh09	4	2399770	216.75	1241
Mesh10	3	3358961	217.75	1249

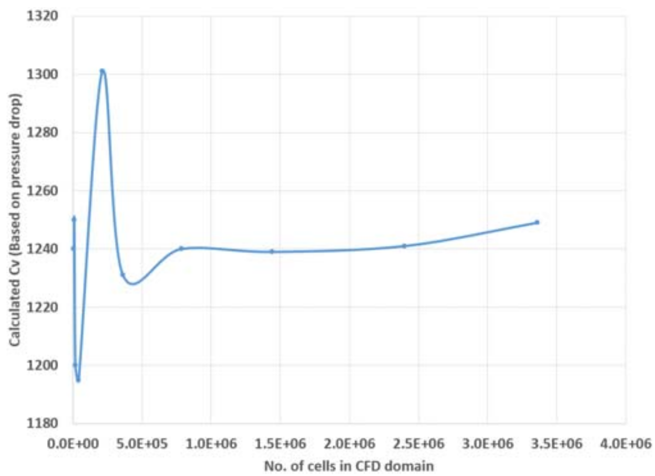

Fig. 8 Mesh size vs CV

The CFD simulations were carried out to study the Cv value of a 3D 12-inch valve using steady state method in ANSYS FLUENT solver. The properties of the fluid are shown in Table

II. Temperature calculations are ignored, and only fluid domain was considered.

TABLE II
FLUID THERMOPHYSICAL PROPERTIES

Property	Water
Density (kg/m^3)	998.2
Viscosity (kg/m-s)	0.001003

IV. RESULTS AND DISCUSSION

Steady state simulations were carried out to predict the flow performance of a control value using CFD simulations. The Cv calculated with the static pressures from the CFD simulation was 1249 which correlated well with Cv of 1300 which is mentioned in the spec sheet of the valve.

Design study was performed to assess the effects of geometrical features which were put with the intent of improving the performance of the valve Cv; four designs were evaluated using CFD by using conditions similar to the baseline case. The change in the static pressure profile because of the geometric features introduced can be observed in Fig. 10, iteration4 shows lower pressure drop across the valve compared to baseline.

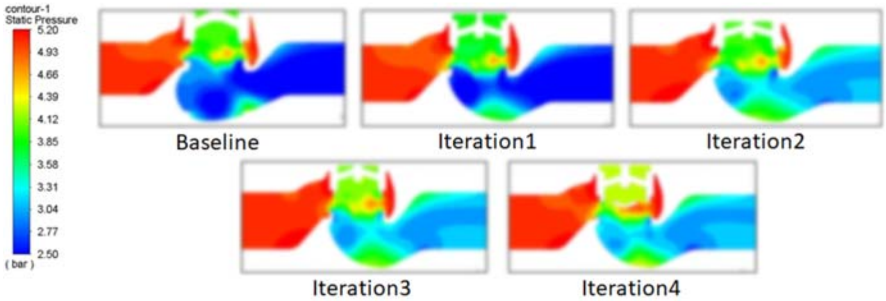

Fig. 9 Static pressure contours for all the designs (Baseline to iteration4)

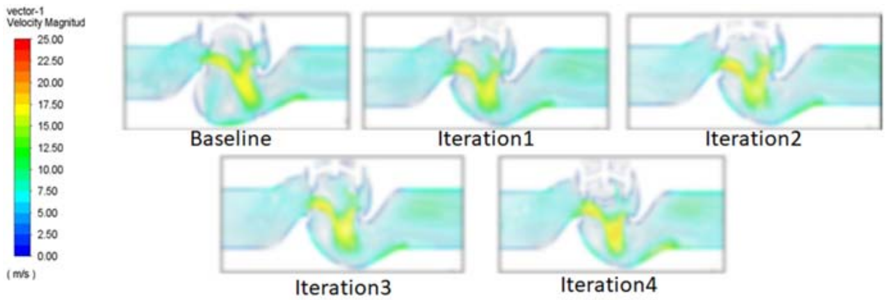

Fig. 10 Velocity vectors for all the designs (Baseline to iteration4)

TABLE III
CV CALCULATIONS

Pressure and Mass flow rate- From CFD simulation	Baseline Design	Iteration1 Design	Iteration2 Design	Iteration3 Design	Iteration4 Design
	Flow to close	Flow to close	Flow to close	Flow to close	Flow to close
Inlet @ 2D	5.01 bar	5.01 bar	5.01 bar	5.01 bar	4.99 bar
Outlet@ 6D	3.01 bar	3.01 bar	3.01 bar	3.01 bar	3.01 bar
Volume flow rate	1531 m^3/hr	1542 m^3/hr	1531 m^3/hr	1512 m^3/hr	1584 m^3/hr
Cv	1249	1264	1241	1236	1301

It can be observed from Fig. 10 that the flow is more streamlined in the iteration1 compared to baseline as the chopped bottom part of the valve is helping in reducing the dead zone to a great extent which was present in the baseline case. Calculated Cv for the iteration1 as well shows the improvement of 15 over baseline. Increasing the cage thickness is leading to higher resistance to flow thereby reducing the valve Cv. Flow structure is getting altered in the top part of the valve in such a way the flow is turbulent which is leading to reduction in calculated Cv for iteraion3. A parabolic plug was introduced just below the cage which is reducing the turbulence observed in top part of the valve to a large extent and making the flow more streamlined, this is evident in the calculated Cv which shows a significant improvement over other designs.

Cv Calculations

Detailed Cv calculation of five models is shown in Table III.

V. CONCLUSION

A few design iterations have been evaluated along with baseline geometry and effects of various design changes on the flow physics and Cv have been assessed within a quick turnaround time. A considerable improvement in Cv value was achieved through design iterations. It has been concluded that addition of parabolic plug (iteration4) has resulted in significant improvement in Cv. Also, this methodology can be used for exploring further options improve the flow and Cv.

REFERENCES

[1] Sreekala, S. K., and U. Muthukumar. "Cv Value Evaluation of a Globe valve using CFD Analysis."
[2] Zhou, Xiao-Ming, Zhi-Kun Wang, and Yi-Fang Zhang. "A simple method for high-precision evaluation of valve flow coefficient by computational fluid dynamics simulation." Advances in Mechanical Engineering 9.7 (2017): 1687814017713702.
[3] Mahajan, Sahil, and Naresh Jaiswal. "Determining Flow Coefficient for Globe Valve with Different Trim shapes using a CFD Tool." International Journal of Engineering Research & Technology (IJERT) 7.10 (2018).
[4] Praveen, J. V. S., and Mehboob Pathan. "Flow coefficient analysis for a globe valve by using CFD." Int J Sci Res 6 (2017): 756-763.
[5] Yevale, Swapnil R., and K. S. Gharge. "A Review on CFD Analysis of Control Valves."
[6] https://www.theprocesspiping.com/introduction-to-globe-valve/
[7] https://kimray.com/training/what-valve-flow-coefficient-cv
[8] https://www.electricalvolt.com/2021/11/measurement-of-control-valve-flow-coefficient-cv/