ISSN: 2455-5703

A Review on CFD Analysis of Control Valves

Swapnil R. Yevale
M.Tech. (Heat Power Engg.)
Department of Mechanical Engineering
Government Collage of Engineering, Karad, Maharashtra, India

K. S. Gharge
Professor
Department of Mechanical Engineering
Government Collage of Engineering, Karad, Maharashtra, India

Abstract

Control valves are generally used as flow control equipment in many industries. The parameter for the performance of control valve analysis is flow coefficient. There is an experimental method to calculate the flow coefficient value of the valve, but the setup for the experimental validation is not readily available as these valves work at high pressure. Due to the progress of the flow simulation and numerical technique (CFD), it becomes possible to observe the flows inside a valve and to estimate the performance of a valve. This paper reviews the researches of the authors in modeling and simulation of the different control valves. The results of this paper provides people with the access of understanding the flow pattern of the valve with different flow rate and also determines the methods which could be adopted to improve the performance of the valve.
Keywords- Valves, Flow Coefficient, Flow Simulation, CFD

I. INTRODUCTION

Control valves are designed to allow precise flow control. They are extensively used for continuous blow off or chemical feed control services. The stem threads are finer than usual so that considerable movement of the hand wheel is required to increase or decrease the opening through the seat. Usually, these valves have a reduced seat diameter in relation to the pipe size. The different types of control valves are as follows,

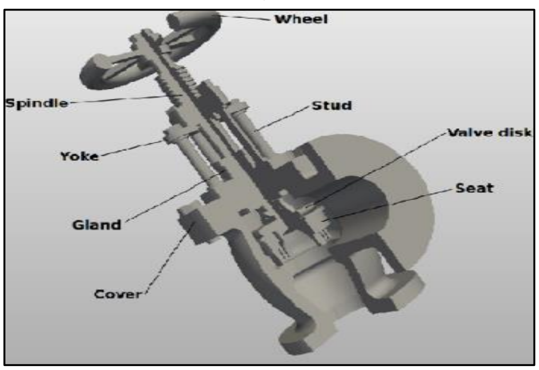

Fig. 1.1: Stop Valve

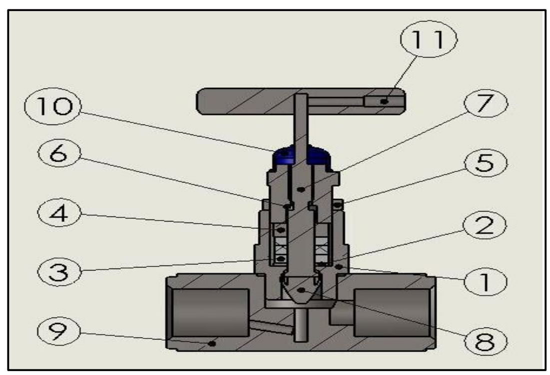

Fig. 1.2: Needle Valve

The flow coefficient of a device is a relative measure of its efficiency at allowing fluid flow. It describes the relationship between the pressure drop across an orifice, valve or other assembly and the corresponding flow rate.
Mathematically the flow coefficient Cv (or flow capacity rating of valve) can be expressed as:

$$Cv = Q\sqrt{\frac{S.G}{\Delta P}} \qquad (1)$$

- 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).

There are several factors on which the Cv value of the valve is dependent; geometry, type of flow, pressure and temperature etc[1]

II. LITERATURE REVIEW

Tansen Chaudhary et al., compares the flow coefficient value at different openings of the valve calculated by ANSYS Fluent 14.5 with the experimental values. Result of the analysis shows reduction in discharge with decrease in opening which satisfies the physics of fluid flow.

All rights reserved by www.grdjournals.com

Brian Nesbitt gave a handbook on valves and actuators which provide the understanding of valves and actuators, properties of fluids (change of state, viscosity, density, compressibility, pH valve, hazards), valve sizing parameters, and serve as a guide for valves installation and maintenance.

Brain Nesbitt classifies needle valve into two types, i.e. Straight needle valve, and angle needle valve.

Anna Budziszewski and Louise Thoren did CFD simulations of a safety relief valve for improvement of one-dimensional valve model in RELAP5. The purpose of their work was to investigate how a safety relief valve can be modelled with CFD and to find parameter relations to be implemented in RELAP5 in order to obtain more realistic results of generated forces in the pipe system.

V. J. Sonawane et al., designed and analysed the globe valve as control valve using CFD software. They analysed globe valves of different sizes and also for different opening conditions. The boundary conditions used were pressure inlet and pressure outlet. They calculated the discharge in every case.

José R. Valdés et al., gave a methodology for parametric modelling of the flow coefficients and flow rate in hydraulic valves. The proposed methodology was based on the derivation from CFD simulations, of the flow coefficient of the critical restrictions as the function of Reynolds number, using a generalized square root function with two parameters.

Hongjun Zhu et al. analysed the flow erosion and flow-induced deformation in needle valve for different inlet rate, opening conditions and different particle sizes. They analysed an angled needle valve, with the help of ANSYS Fluent. The boundary conditions used were velocity inlet and pressure outlet. They analysed the valve for 20, 30, 40 m/s inlet speed and for different opening sizes. Total seventeen permutations were taken for analysing the effect on the valve. The conclusion from the analysis was that the effect of particle diameter on erosion is most significant followed by inlet rate.

Rodrigo Alvite Romano and Claudio Garcia. describes two methods used to estimate the parameters of a Karnopp friction model applied to control valves. The methods are tested using simulated data of three valves with different friction levels.

III. GOVERNING EQUATIONS

A. *Continuity Equation*

The continuity equation is formulated as

$$\partial\rho/\partial t + \partial\rho U_i/\partial x_i = S(\rho) \qquad (2)$$

Where the first term is the rate of accumulation of mass, the second term is the transport of mass by convection and the third term the source term. The subscript i ranges from 1 to 3 in 3D. for one phase, incompressible flow, i.e. a Mach number < 0.3, the source term and the accumulation term can be excluded which results in the equation

$$\partial U_i/\partial x_i = 0 \qquad (3)$$

B. *Momentum Equation*

The momentum equation, also known as the Navier-Stokes equation, is written as

$$\partial U_i/\partial t + U_j\partial U_i/\partial x_j = -(1/\rho)\partial P/\partial x_i + (1/\rho)\partial\tau_{ji}/\partial x_j + g_i \qquad (4)$$

Where the first term on the left-hand side describes the accumulation of momentum and the second term describes the convective acceleration. On the right-hand side the terms describe the pressure forces, shear forces and gravity force.

IV. CFD ANALYSIS

CFD analysis consists of the following steps: creating discrete models of flow paths, adjusting simulation parameters, carrying out simulations and analyzing the obtained results.

A. *Creating Discrete Models of Flow Paths*

Discrete models of the flow paths are created in ANSYS/Fluent on the basis of geometric models. The geometric models can be created using software like CATIA, SOLIDWORKS, PRO-E etc. The numerical grid for the valve are generated by means of Gambit, a pre-processor of Fluent.

The grid has a significant impact on Rate of convergence (or even lack of convergence), Solution accuracy CPU time required. Mesh quality for good solutions are Grid density, adjacent cell length/volume ratios, Skewness, Tet vs. hex, Boundary layer mesh, Mesh refinement through adaption.

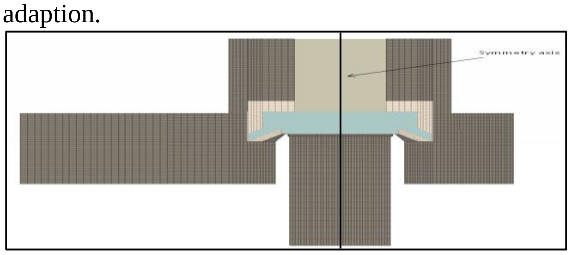

Fig. 4.1: Meshing of safety relief valve

All rights reserved by *www.grdjournals.com*

Fig. 4.2: Meshing of needle valve

B. Simulation Parameters

Before the CFD analyses were carried out, the type of flow was determined in order to apply the turbulence model. ANSYS/Fluent provides a number of turbulence models, including k−ϵ, k−Ω and Reynolds. In the case of flow through the channels of a control valve, the k−ϵ model fits well. The inlet velocity magnitude or inlet mass flow rate set in the Boundary Conditions option as normal to the boundary. The Velocity magnitude or mass flow rate value is introduced as the Workbench input parameter. Outlet pressure is chosen in the Outlet condition option.

C. Results of CFD Simulations

Results are the objective of every CFD simulation. They reveal the performance of design. It shows whether the design meets satisfies its objectives or not. Results are essential for making informed design decisions. In CFD Results are in terms of Contours, Streamlines, Tracing, Tracking, Volumetric rendering, Vectors, of various parameters like Velocity, Pressure, Density, Temperature, Turbulent Kinetic Energy, Dissipation rate etc.

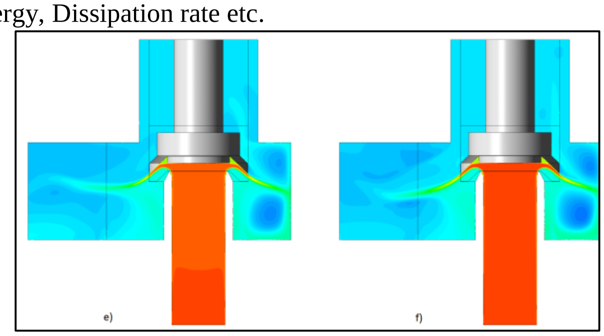

Fig. 4.3: Pressure contour of safety relief valve

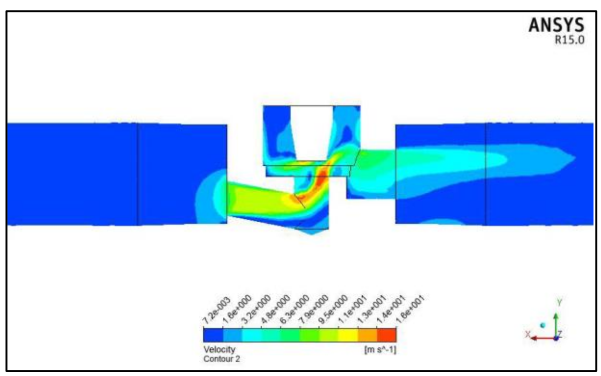

Fig. 4.4: Velocity contour for needle valve

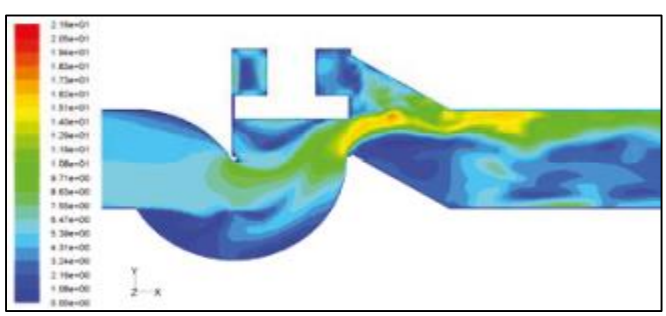
Fig. 4.5: Velocity contour for stop valve

All rights reserved by www.grdjournals.com

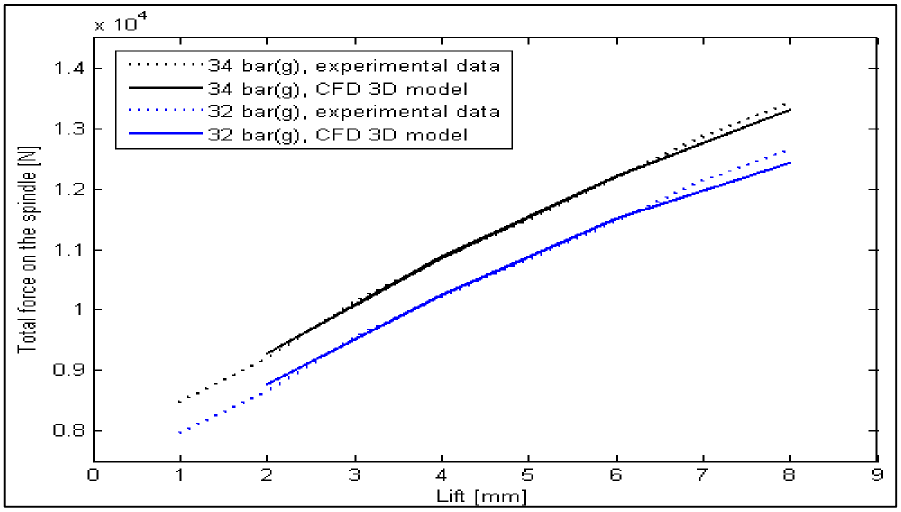

Fig. 4.6: Total forces acting on the spindle for different magnitudes of lift during steady state simulations.

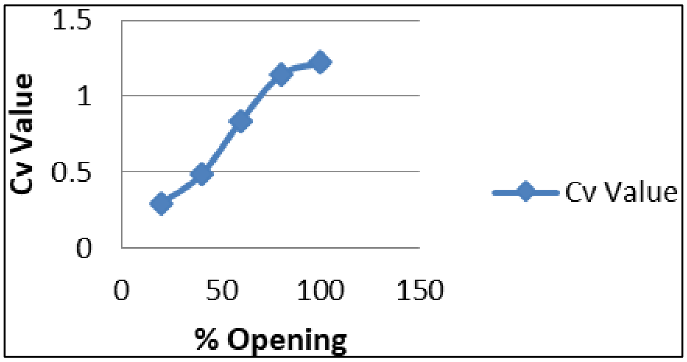

Fig. 4.7 Variation in flow coefficient with percentage opening

V. Conclusion

The simulation results of stop valve shows that the main pressure drop is generated along the throat path. Flow jet produced by the valve exit of the throat can be easily distinguished. Needle valve study covers the performance of valve on the basis of flow coefficient values. It is found that the maximum error for coefficient of flow is 6.05% which is within an acceptable range. The CFD simulations in 2D and 3D show similarities, such as the opening time of the valve and the order of magnitude of hydraulic forces, when the inlet pressure is instantly increased. However, some differences are evident between the models, such as the built up back pressure and the opening process during gradual increase of inlet pressure.
The results of the CFD analysis of the control valve show nearly same results with the experimental results.

References

[1] Romil Khowal, D.N. Jadhav, Tansen Chaudhari “Flow Coefficient Valve Calculation using CFD Analysis for Needle Valve” Special Issue on International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME) V-5 No.2 ISSN (Print): 2319-3182, For National Conference on Advances in Design and Thermal Engineering (NCADTE-2016) G.H. Raisoni College of Engineering & Management, Chass, Ahmedanagar, Maharashtra 17th to 18th February 2016
[2] Brian Nesbitt, Handbook of valves and actuators.Elsevier,17
[3] Louse Thoren, Anna Budziszewski, "CFD simulation of a safety relief valve for improvement of a one-dimensional valve model in RELAP5," Chalmers University of Technology, 2012
[4] T. J. Rane, A.D. Monde, R.V. Vajarinkar, P. C. Gawade V. J. Sonawane, "Design and Analysis of Globe Valve as Control Valve Using CFD Software," IOSR Journal of Mechanical and Civil Engineering, pp. 63-71
[5] José M. Rodríguez, Javier Saumell, Thomas Pütz José R. Valdés."A methodology for the parametric modelling of the flow coefficients and flow rate in hydraulic valves,"Elsevier, vol. 88, pp. 598-611, December 2014
[6] Qian Pan, Wenli Zhang, GuangFeng, Xue Li, Hongjun Zhu, "CFD simulations of flow erosion and flow-induced deformation of needle valve: Effects of operation, structure and fluid parameters.," Elsevier, vol. 273, pp. 396-411, July 2014
[7] E.Lisowski, G.Filo “Analysis of proportional control valve flow coefficient with the usage of the CFD method” Instrumentation Volume, March 2017, Pages 269-278 [14] Taewoo Kim, Sulmin Yang, Sangmo Kang. Numerical study on the flow characteristics of a solenoid valve for industrial applications. Wseas Transactions on Fluid Mechanics; July 2010, Issue 3, Volume 5; p. 155-164.
[8] Fluid Controls Pvt. Ltd., Pune Catalouge.
[9] Qin Yang,Ziguo Zang,Mingyue Liu,Jing Hu “Numerical Simulation of Fluid Flow inside the Valve” Procedia Engineering 23 (2011) 543 – 550.

All rights reserved by *www.grdjournals.com*