Flow Coefficient Valve Calculation using CFD Analysis for Needle Valve.

[1]Romil Khowal, [2]D N Jadhav, [3]Tansen Chaudhari

[1,2]SPCE, Mumbai, [3]Fluid Controls Pvt. Ltd., Pune
Email: [1]romilkhowal@gmail.com, [2]d_jadhav@spce.ac.in, [3]tansen.chaudhari@fluidcontrols.com

Abstract— Needle valves are widely used in instrumentation industries. The performance parameter for needle 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. This paper compares the flow coefficient value at different openings of the valve calculated by ANSYS Fluent 14.5 with the experimental values. The purpose of this paper is to define the boundary condition for the CFD analysis of the valve and to reduce the experimental validation for different sizes and openings and hence reducing the overall expense. Result of the analysis shows loss in discharge with reduction in opening which is in line with the physics of fluid flow.

Index Terms—CFD, Cv value, Flow Coefficient, Needle Valve.

I. INTRODUCTION

Needle valve, having a small port and a needle shaped plunger are widely used to get precise flow [1]. The change in direction in the flow through needle valve introduces changes in the fluid flow. The needle in the valve is hardened to withstand the flow pressure and to avoid corrosion and erosion, ensuring precise fluid flow. These valves are generally used for low flow rates and have a preferred direction of flow.

Parcol's handbook for control valve sizing defines flow coefficient (Cv value) as the standard flow rate which flows through a valve at a given opening i.e. referred to the following conditions:

- Static pressure drop (ΔP(Cv)) across the valve of 1 psi.

- Flowing fluid: water at a temperature from 40 to 100o F

- Volumetric flow rate: expressed in gpm [8].

$$C_v = q_v \times \sqrt{\frac{\nabla p_{C_v}}{\nabla p} \times \frac{\rho}{\rho_o}} \qquad (1)$$

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

Hongjun Zhu et al., performed CFD simulations to understand the effects of operation, structure and fluid parameters on flow erosion and flow induced deformation of needle valve. José R. Valdés et al., proposed a methodology for parametric modelling of the flow coefficients and flow rate in hydraulic valves. In recent days, Computational Fluid Dynamics (CFD) has been experiencing rapid advances due to both computer technology progress and efficient algorithms that have been developed to solve the Navier-Stokes (N-S) equations used in the flow analysis around ship hulls as discussed by J. B. V. Wanderley et al..

II. PROBLEM DEFINITION

Fig. 1 shows the sketch and bill of materials of needle valve (model number: 8 NVFN SS315) investigated in this study. Ten cases are modeled and analyzed in this paper. Case 1 is with 100% opening and case 10 as 10% opening.

III. GOVERNING EQUATIONS

Following equations are considered for fluid flow [2]

1. Mass conservation equation or Continuity Equation

$$\nabla \times (\rho V) = 0 \qquad (2)$$

Where,

ρ= density of fluid, V= velocity of fluid

2. Momentum Equation or Navier-Strokes Equation

$$\nabla \times \left(\rho V V - \tau_f\right) = f_f \qquad (3)$$

Where,

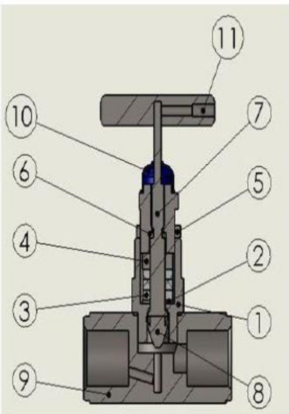
$$\tau_f = \left[-P + (-g - \frac{2}{3}\mu)\nabla \times V\right] I + 2\mu E$$

$$E = \frac{1}{2}(\nabla V + \nabla V^T)$$

τ_f = Stress of fluid, f_f = Volume force of fluid, P= Pressure, g= Gravitational acceleration, μ = Dynamic viscosity, I = Second order unit tensor.

IV. NUMERICAL METHOD

Computational flow dynamics is used to solve the Navier-Strokes equations. To perform CFD analysis of any component solid model of the same is required. The model of different openings is modeled in SOLIDWORKS. CFD simulations are done in ANSYS FLUENT 14.5. The Flow domain as shown in Fig. 2 is extracted from ANSYS Design Modeller 14.5. The fluid domain is divided in discrete cells called mesh with the help of ANSYS MECHANICAL 14.5 as shown in Fig. 3. Six different girds are used to achieve the grid independence for case 1. In general practice, it is preferred to take Tetrahedron mesh type for computational flow dynamics[2]. For the connector between pipe and inlet of valve, automatic method is preferred with an element size of 1 mm. For the valve portion, patch conforming method is used with an element size of 0.5 mm, as it a good compromise between computational time and result accuracy. The mesh generation parameters and the pressure drop are plotted in Fig. 4. The boundary conditions applied are Inlet pressure as 40 bar and outlet conditions as 39 bar. As the flow in the valve is considered turbulent, realizable k-epsilon model is considered for the analysis [3]. The results from ANSYS fluent are tabulated in table 1.



Item No.	Part	Qty.
1	Bonnet	1
2	Gland seat washer	1
3	Gland seal	3
4	Gland retainer	1
5	Check nut	1
6	Gland	1
7	Spindle	1
8	Spindle plug	1
9	Body	1
10	Dust cap	1
11	Handle	1

Fig. 1: Needle Valve and Bill of Materials

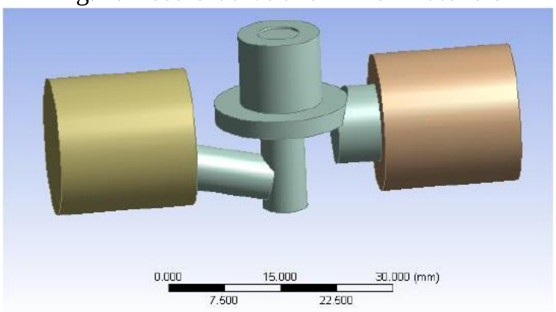


Fig. 2: Fluid Domain

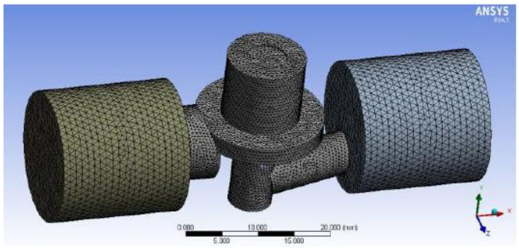


Fig. 3: Meshing

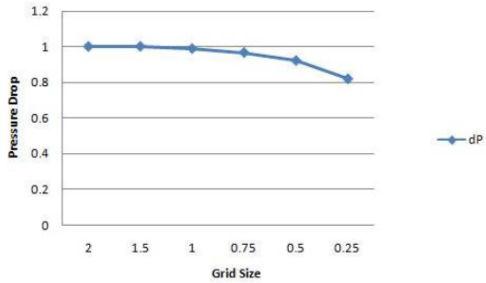


Fig 4: Grid Independence test

V. RESULT AND DISCUSSIONS

The Analysis is run for ten opening conditions. The results of the same is listed in the below table. The Variation of pressure drop is near to the applied pressure condition in each case. Other Parameters shows expected behavior as per the physics of the valve.

Case 4 and Case 9 does not follow the pattern of the valves. The reason of the same is the opening of the valve, in both cases needle supports the fluid flow hence result in increase in outlet velocity.

$$q_v = A \times V \qquad (4)$$

Experimentally tested Cv value for 100% open condition of the tested valve is around 4.75.

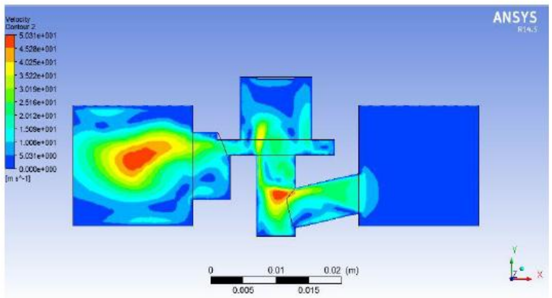


Fig. 5: Velocity contour for 100% open case

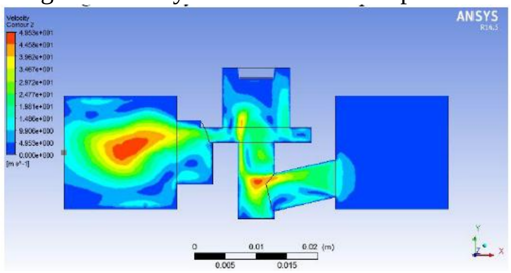


Fig. 6: Velocity contour for 90% open case

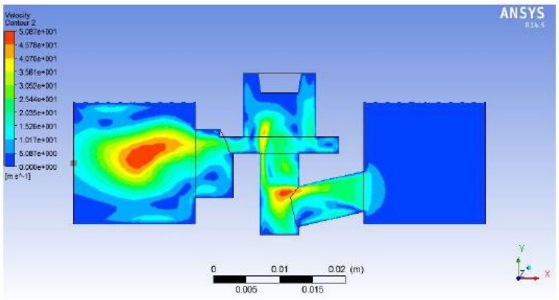


Fig. 7: Velocity contour for 80% open case

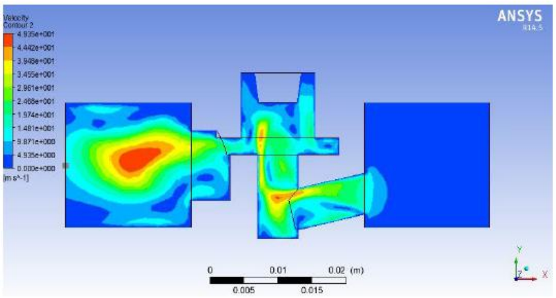


Fig. 8: Velocity contour for 70% open case

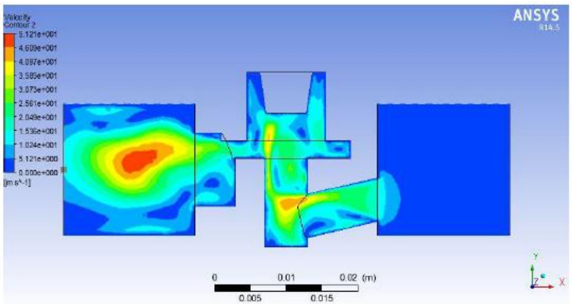


Fig. 9: Velocity contour for 60% open case

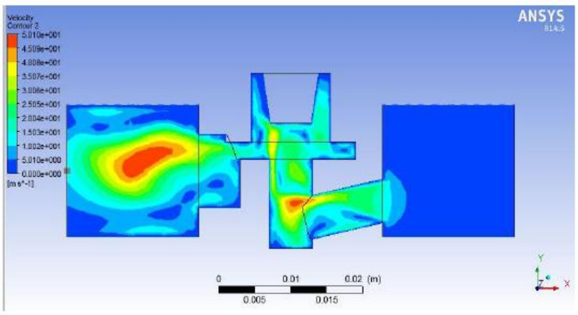


Fig. 10: Velocity contour for 50% open case

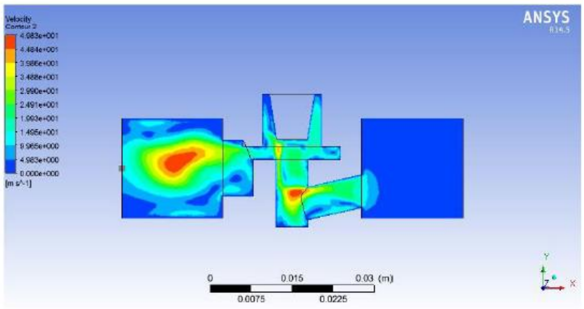


Fig. 11: Velocity contour for 40% open case

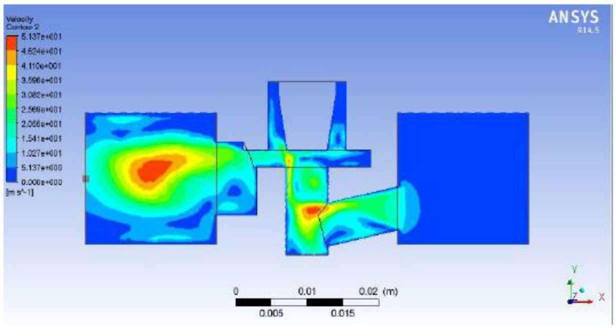


Fig. 12: Velocity contour for 30% open case

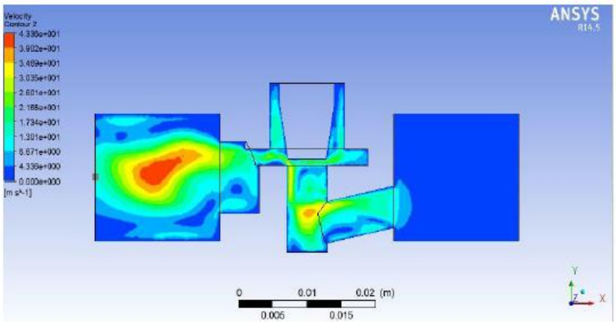


Fig.13: Velocity contour for 20% open case

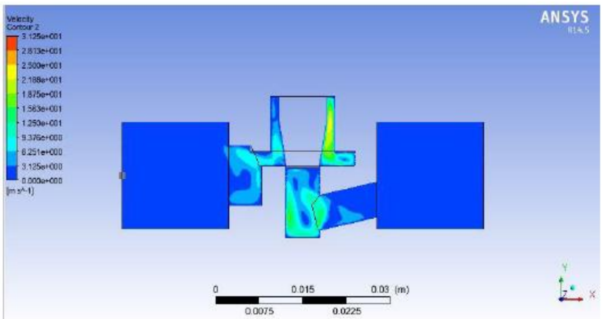


Fig. 14: Velocity contour for 10% open case

I. Parameters at different openings

S. No.	Opening (%)	Outlet Velocity (m/s)	Pressure Drop (bar)	Flow Coefficient	
				K_v	C_v
1	100	5.09222	1.01385	4.80107394	5.550041474
2	90	4.71247	1.00952	4.452554292	5.147152761

3	80	4.624	1.00603	4.376535429	5.059274956
4	70	4.54271	1.00489	4.302034013	4.973151319
5	60	4.60426	1.00366	4.362994052	5.043621124
6	50	4.70054	1.0083	4.443968358	5.137227422
7	40	4.47733	1.00592	4.237946526	4.899066184
8	30	4.75354	1.00818	4.494342885	5.195460375
9	20	4.99131	1.02656	4.676710252	5.406277051
10	10	0.372812	0.99982	0.353954193	0.409171047

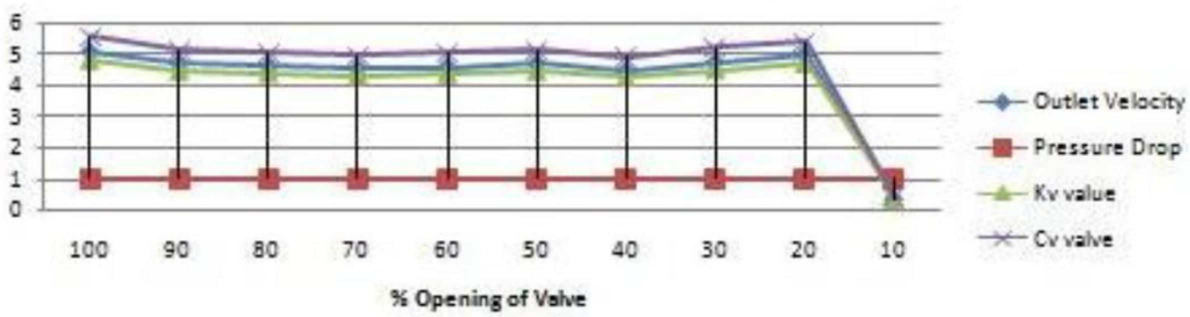


Fig. 15: Parameter vs % Opening curve

VI. CONCLUSION

The variation in experimental value and the value from numerical method is about 14.41%. In general practice up to 20% variation is considered okay. Thus the boundary conditions used in this paper are within permissible limits.

REFERENCES

[1] Brian Nesbitt, Handbook of valves and actuators.: Elsevier, 2007.

[2] Qian Pan, Wenli Zhang, Guang Feng, Xue Li, Hongjun Zhu, "CFD simulaions of flow erosion and flow induced defromation of needle valve: Effects of operation, structure and fluid parameters.," Elsevier, vol. 273, pp. 396-411, July 2014.

[3] ANSYS, "ANSYS Fluent User Guide," User Manual 2014.

[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] LOUISE 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.

[7] Swagelok. Swagelok. [Online]. http://www.swagelok.com/downloads/animations/en/flashapps/swageloku/SPROD-01SNIP/player.html

[8] Parcol, Handbook for control valve sizing. Canegrate MI, Italy: Parcol SPA.

[9] FTI HAM-LET Catalogue. ftitechinfo.com. [Online]. http://www.ftitechinfo.com/valvesandfittings/needlevalves/index.html

[10] Parker Hannfin Corporation. forberg.com. [Online]. http://www.forberg.com/pdf/techSup/Home_Tech%20Support_Valve%20Flow_calc_and_sizing.pdf

[11] International Society of Automation, "Flow Equations for Sizing Control Valves," ISA, North Carolina, Standard ISA-75.01.01-2007, 2007.

◈◈◈