Proceedings of the 37th International & 4th National Conference on Fluid Mechanics and Fluid Power

FMFP2010

December 16-18, 2010, IIT Madras, Chennai, India

FMFP2010_ 341 ___

MODELING AND SIMULATION OF VALVE COEFFICIENTS AND CAVITATION CHARACTERISTICS IN A BALL VALVE

Yogesh Gawas
Sardar Patel College of Engineering.
Mumbai, Maharashtra, India
g15engineering@yahoo.co.in

Dr. Vilas R. Kalamkar
Sardar Patel College of Engineering.
Mumbai, Maharashtra, India
vilas.kalamkar@gmail.com

Vijay Mali
Director, Centre for Computational Technologies (CCTech) Pvt. Ltd.
Pune, Maharashtra, India.
vijay@cctech.co.in

ABSTRACT

Ball valve, a rotational motion valve uses ball shaped disk. It is used for on/off or throttling operations. It offers the minimum resistance to flow. To investigate the valve performance and its characteristics, the flow through the valve is studied using numerical technique. In this paper the numerical simulations were performed using commercial code FLUENT, to study the flow patterns and to estimate the valve sizing coefficient, torque coefficient and cavitation index for investigation of cavitating flows. These simulations were performed at different pressure drops and for varying percentage opening of the valve.

Keywords: Ball Valve, Cavitation, Numerical Simulation, Valve Coefficients.

INTRODUCTION

Ball valve is a quarter-turn valve that features a spherical closure device. As the ball move radially across the seal, the opening in the ball is exposed, which allow the flow. Ball valve is also categorized as high-pressure recovery valve. At intermediate openings, there are two throttling ports in series, one at the inlet and other at the outlet of the ball. Hence the system experiences double pressure drop, due to which ball valve has better cavitation characteristics.

Computational Fluid Dynamic (CFD) technique, are much developed and are used as an important tool in scientific research and also in industrial engineering designs. By now, the CFD simulation by commercial software had proved their ability to predict the flow characteristics. The flow pattern inside the valve, formation of vortices, and complex phenomenon like cavitation can be visualized by CFD techniques.

CFD has become a popular tool for design of fluid machinery, valves, heat exchanger and many other devices. There have been many reports on valves, in which different flow phenomena were analyzed using CFD technique. The commercial code, STAR-CD^{TM}, was used to investigate fluid flows through a ball valve and to estimate the important coefficients by Chern and Wang, 2004. Using FLUENT^{TM}, investigation of flow around a V-sector ball valve was performed by Merati et al., 2001. A three dimensional numerical analysis, was performed to reveal the velocity field, pressure distributions in a butterfly valve by using FLUENT^{TM} by Huang and Kim, 1996. Using AVL-Fire^{TM}, the flow containing the bubbles in a ball valve was analyzed by van Lookeren Campagne et al., 2002. To perform three dimensional analysis, to estimate pressure drop, flow coefficient and hydrodynamic torque coefficient in a butterfly valve ANSYS CFX^{TM} was used by Song and Park, 2007. Implementing mixture model of FLUENT^{TM}, the cavitating turbulent flow for two dimensional NACA0009 hydrofoil was analyzed by Bernad et al., 2006.

In this research the main objective is to model the fluid domain of 10 inch ball valve, along with the prescribed length of upstream and downstream piping system. Commercial package ICEM-CFD 12.0 was used as pre-processing tool, while FLUENT 12.0 was used as solver and for post-processing. FLUENT provides a three dimensional numerical simulation of water through the ball valve and the fluid domain, and also helps to estimate the pressure drop, volume flow rate, sizing coefficient, torque coefficient, and cavitation index.

Fig. 1. Trunnion type full port ball valve

VALVE SIZING COEFFICIENT, INHERENT CHARACTERISTICS, TORQUE COEFFICIENT, AND CAVITATION INDEX

Valve sizing coefficient (Cv)

It is defined as quantity of water in US gallons at 60^{o}F that will pass through the valve each minute with a 1 psi pressure drop across it. It is also a measure of capacity of valve, which takes account of its size and natural restriction to flow through the valve. Also it is a dimensional value. It can be calculated by Eq. (1).

$$Cv = 1.16 \times Q \times \sqrt{\frac{S.G}{\Delta P}} \quad (1)$$

Torque coefficient (Ct)

Forces required to open/close a quarter-turn valves are caused by friction and hydrodynamic forces. Friction forces act at the valve seat and bearing surfaces. Hydrodynamic torque is caused by forces induced by the flowing fluid. Torque coefficients are dimensionless entities, varies with valve opening position. It is calculated by the Eq. (2).

$$Ct = \frac{T}{D^3 \times \Delta P} \quad (2)$$

Cavitation Index (CI)

Cavitation Index is a parameter derived from the dimensional analysis, which corresponds to the intensity of cavitation. It is defined as the ratio of forces trying to suppress cavitation to the force trying to cause it. For valves and other devices that create a pressure drop, Cavitation Index can be further defined in several ways as explained in Fig. (2). Pressure Tab 1 is installed one diameter upstream from valve. Pressure Tab 2 is installed five diameters downstream from valve, as shown in Fig. (2). If the reference pressure in numerator is upstream pressure, P_{t1} then

$$CI = \frac{P_{t1} - P_{sat}}{\Delta P} \quad (3)$$

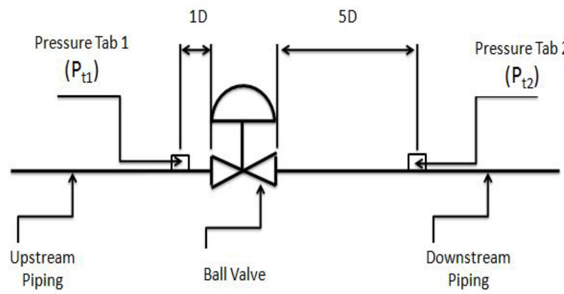

Fig. 2. Schematic diagram of ball valve and piping system

If the reference pressure in numerator is downstream pressure, P_{t2} then

$$CI_1 = \frac{P_{t2} - P_{sat}}{\Delta P} \quad (4)$$

Equation (4) is a much preferred form over (3), since downstream pressure is the pressure closer to zone, where cavitation actually occurs. Therefore downstream pressure, more directly influences the cavitation process. Equation (3) & (4) are related by the following Eq. (5).

$$CI = CI_1 + 1 \quad (5)$$

CFD MODELLING

Model description

For this study 10 inch nominal diameter ball valve geometry is used. The required fluid domain was extracted and the prescribed length of piping system was added. Accuracy of simulation mainly depends on quality of grid, hence to get better results, tetrahedral mesh generated using ICEM-CFD 12.0, were converted to polyhedral mesh using FLUENT 12.0.

Numerical approach

Basic Flow Model: Working fluid is water, hence incompressible and viscous fluid flows through the ball valve. Pressure drop between inlet and outlet being very high induces high velocity flow, hence flow studied is turbulent in nature.

Turbulence Model: To capture turbulence, Reynolds Averaged Navier-Stokes (RANS) equation is utilized. Its common form is written as in Eq. (6);

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_i}(u_i u_j) = -\frac{\partial p}{\partial x_i} + \quad (6)$$

$$\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u'_i u'_j}\right)$$

Where u is the mean velocity and the subscript, *i, j=1~3*, refers to Reynolds-averaged components in three directions respectively. These Reynolds Stresses, $-\rho\overline{u'_i u'_j}$, must be modeled in order to close the equation. For this study realizable k-ε turbulence model is utilized.

Cavitation Modeling: When local static pressure as some points falls below vapor pressure, bubbles are formed. If this pressure recovers downstream then bubbles collapses, which leads to cavitation, if pressure does not recover bubbles are entrained along with the flow, this process is called flashing, which is more severe compared to cavitation. These process are shown in Fig. (3). The term nuclei is other name for gas bubbles. For cavitation/flashing to occur, there must be nuclei present. If liquid was completely deaerated (i.e. no nuclei), then liquid will cavitate, far below the normal liquid vapor pressure.

Vapor Transport Equation: Working fluid is assumed to be a mixture of liquid, its vapor and non-condensable gases. Standard governing equations in Mixture Model and Mixture Turbulence Model describe the flow and account for the effects of turbulence. Vapor transport equation governs the vapor mass fraction 'f', given by Eq. (7).

$$\frac{\partial}{\partial t}(\rho_m f) + \nabla(\rho_m \vec{v}_v f) = \nabla(\gamma \nabla f) + R_e - R_c \quad (7)$$

Where ρ_m = Mixture density,

$\vec{v}_v$ =velocity vector of vapor phase,

γ =effective exchange coefficient,

R_e & R_c = vapor generation and condensation rate terms, derived from Rayleigh-Plesset equations and limiting bubble size consideration.

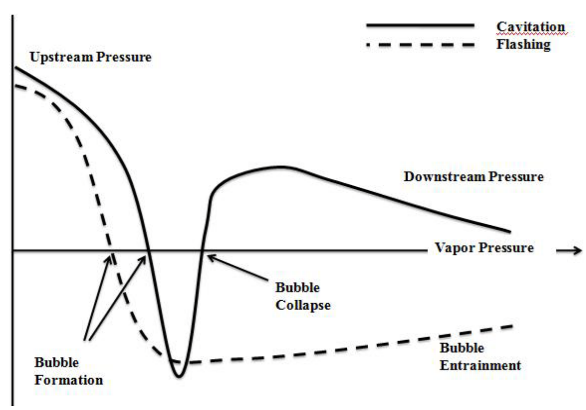

Fig. 3. Pressure variation for cavitation and flashing process

Working Fluid, Boundary Conditions

Table. 1 show the working fluid used in analysis, type of boundary conditions used, phases involved.

Table. 1: Boundary condition and other parameters

For Numerical Prediction of		
	Valve sizing and Torque Coefficient	Cavitating Flow
Working Fluid	Water	Water, Water Vapor, Non-Condensable Gases
Phases	Single Phase Flow	Two Phase Flow (Water, Water Vapor)
Inlet Boundary Condition	Pressure Inlet Boundary Condition	Pressure Inlet Boundary Condition
Outlet Boundary Condition	Pressure Outlet Boundary Condition	Pressure Outlet Boundary Condition

RESULTS AND DISCUSSION

Simulations were performed at different pressure drops and varying percentage opening of valve. Fig. (4) shows the velocity contour plot for 60 percent opening of valve.

Fig. (5) shows Valve Sizing Coefficient graph. The experimental value of Cv for 100 percent opening is 3180 [8]. The value obtained from simulation result for 100 percent opening is 3086. Hence experimental Cv values and numerically obtained values can be compared for validation.

Hydrodynamic torque reduces with pressure drop. As valve is closed hydrodynamic torque increases to a maximum value and then reduces. Torque coefficient remains constant even as pressure varies. As valve is closed torque coefficient increases to a maximum value and then reduces. Fig. (6) shows the graph of variation in torque coefficient.

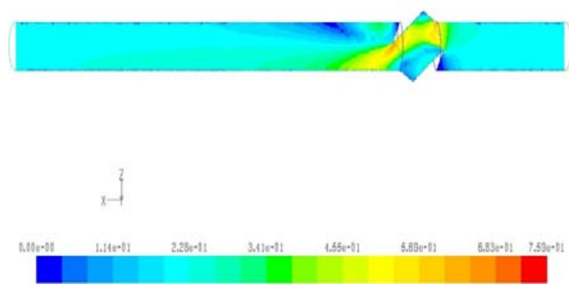

Fig. 4. Velocity contour plot for 60 percent opening

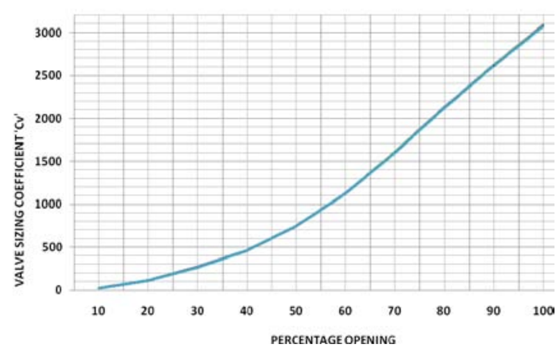

Fig. 5. Graph of variation of valve sizing coefficient

Fig. (7) shows graph of absolute pressure variation for 30 percent opening of valve. Fig. (8) shows contour plot for velocity of mixture (water, its vapor and non-condensable gases) for 30 percent opening. Fig. (9) shows the contour plot of volume fraction of vapor phase for 30 percent opening, it also indicates the cavitating zones. Fig. (10) shows graph of cavitation index for 10 percent opening and varying pressure drop.

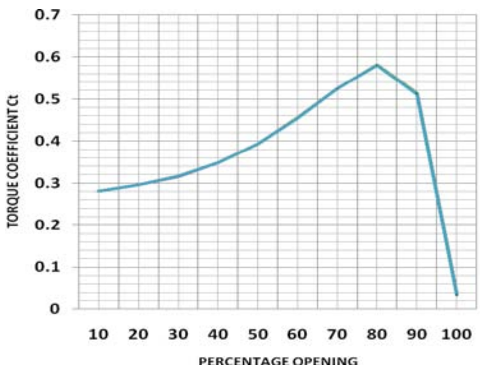

Fig. 6 Graph of variation of torque coefficient

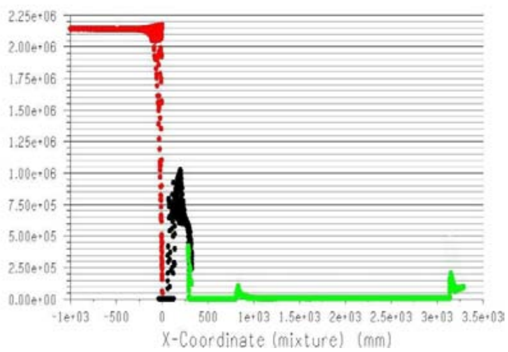

Fig. 7. Graph of absolute pressure variation for 30 percent opening

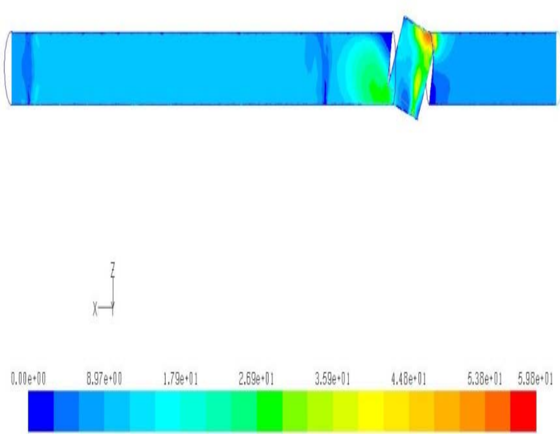

Fig. 8. Mixture velocity contour plot for 30 percent opening

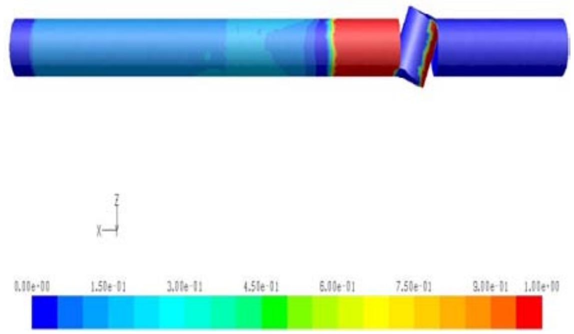

Fig. 9. Volume fraction of vapor phase for 30 percent opening

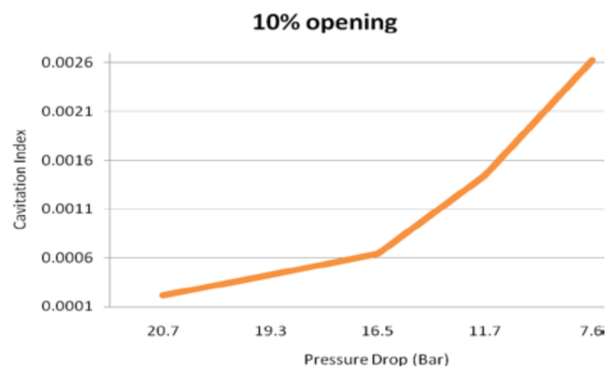

Fig. 10. Cavitation index for 10 percent opening and varying pressure drop

CONCLUSIONS

1. Valve sizing coefficient and Torque coefficient depends on geometry and not on the flow conditions.
2. Maximum hydrodynamic torque occurs at 60% opening, while maximum torque coefficient occurs at 80% opening. Though maximum torque coefficient is obtained at 80 percent of valve opening, it does not mean that hydrodynamic torque is maximum at the same percentage opening
3. Cavitation Index depends on geometry as well on flow conditions. As valve is closed pressure drop across it increases,

thus the cavitation index decreases. Hence valve operating at lower percentage opening cavitate severely.

4. Hence while designing the valve systems designer must consider the entire range of sizing coefficient, hydrodynamic torque and torque coefficient and cavitation index.

NOMENCLATURE

Symbol	Quantity	Units
D	Nominal Diameter of Ball Valve	mm
f	Mass Fraction	
Q	Volume Flow Rate	m^3/s
$S.G$	Specific Gravity	
T	Hydrodynamic Torque	N-m
u	Mean Velocity	m/s
$\vec{v}$	Mass Averaged Velocity	m/s
α	Volume Fraction	
γ	Effective exchange coefficient	
μ	Viscosity	Kg/m-s
ρ	Density	Kg/m^3
ΔP	Pressure Drop	Pa

REFERENCES

❖ ***Journal articles***

Chern, M. J., Wang, C. C., 2004. Control of volumetric flow-Rate of ball valve using V-port, Journal of Fluid Engineering 126 471–481.

Merati, P., Macelt, M. J. and Erickson, R. B., 2001. Flow investigation around a V-sector Ball Valve, Journal of Fluid Engineering 118 662-671.

Huang, C. and Kim, R. H., 1996. Three-dimensional analysis of partially open butterfly valve flows, Journal of Fluid Engineering 123 562-568.

van Lookeren Campagne, C., Nicodemus, R., de Bruin, R. J. and Lohse, D., 2002. A method for pressure calculation in Ball valve containing bubbles, Journal of Fluid Engineering 124 765-771.

Song, X. and Park Y. C., 2007. Numerical analysis of butterfly valve-prediction of flow coefficient and hydrodynamic torque coefficient, World Congress on Engineering and Computer Science.

❖ ***Books***

Skousen, P. L., 1997. Valve Handbook, first ed. McGraw-Hill, pp. 221-478.