Flow Evaluation of Ball Valve for Performance Enhancement Using CFD Software

Vishal Andhale¹, Dr. D. S. Deshmukh ²

¹ PG student in mechanical engineering department, SSBT’s, COET, Jalgaon, India.
² Professor and Head of mechanical engineering department, SSBT’s, COET, Jalgaon, India.

Email: vish8501@gmail.com
deshmukh.dheeraj@gmail.com

Abstract: Ball valves are commonly used as flow control device, in applications where the discharge pressure required from the valves are relatively low. It is observed that design the valve in such a way that it will give best performance and optimum efficiency would be achieved in hydraulic power plants. Owing to fast progress in the development of flow visualization and numerical technique availability, it is possible to observe the flow around a valve and to estimate the performance of a valve. In this study the flow evaluation through 1 ½” CPVC Ball valve is carried out using computational Fluid Dynamics (CFD) software. Modeling of the valve is done using CATIA V5 R20 software and for analysis ICEM CFD 12 is used. Valve parameters including flow coefficient and head loss coefficient have been considered using CFX 12. The fluid is taken as incompressible fluid for design and analysis of valve. Magnitude of head loss coefficient calculated from numerical analysis is compared with the experimental data and the results are found satisfactory.

Keywords: CPVC Ball Valves, Optimum Performance, ICEM CFD, CATIA V5 R20

I. Introduction: Ball valve is high quality; economical quarter-turn shutoff valve designed for irrigation, pool and spa, and general purpose applications. It has simple mechanical meeting and small flow resistance in a totally open function. It provide a surprisingly excessive flow capacity. They’re most suited for exceptionally low pressure with the flow. Usually, the fluid friction coefficient is low and additionally the increase is normally minimal because of the Ball valve is operated with a quarter turn. Manual operation may be through lever or tools provided with it. The primary try at gathering and collating the published statistics regarding Ball valves was likely made via Cohn in 1951. Experimental studies on Ball valve go with the flow traits are performed by way of Addy et al in 1985. The outcomes of numerical simulation of go with the flow characteristics inclusive of each velocity and pressure calculations are offered in literature. Examine on hydrodynamic torque of the Ball valve are conducted by means of Morris and Dutton in 1989.

Ball valves are required to have excessive performance characteristics and higher precision as they are used as close off valves. The characteristics of a valve i.e. head loss characteristic, torque traits and force characteristics of Ball valve is decided conventionally through tests. If take a look at valve is of large size, scale version of valve is examined to decide its traits. Evaluation of flow feature experimentally is a hectic and is not very precise work. Exact theoretical evaluation of flow through complex geometry may be very hard with using high velocity computers, and the numerical techniques, the flow evaluation can be made the use of CFD. So flow analysis is to be performed using simulation software. Goal of this research work is to find amount of water leakage from ball valve at closed condition. At present nearly each enterprise is using software for evaluation. Ball valves are used in various power plants and it has extensive scope. So this study will be beneficial for all fields where Ball valves are used. Computational fluid dynamics is a tool to perform numerical solution with the flow simulation for predicting the float behavior within a particular domain via numerical answer of governing equations to ideal accuracy. Computational fluid dynamics is becoming very beneficial approach for engineering layout and analysis due to improved numerical method and at the same time, it saves time and power of experimental setups.

II. Literature Review

A series of numerical CFD simulations of the flow through ball check valve of an ABS control valves are performed, with and without inclusion of a cavitating model, under similar conditions of pressure difference between inlet and outlet ports by Jose R. Valdes. [1] And concludes that the correlation between measurements and CFD predictions is excellent in both cases, thus validating the accuracy of simulations and cavitating model for valve flow prediction. The significant difference between cavitating and non-
cavitating conditions shows importance by taken into account the cavitation effects in prediction of the flow rate.

A methodology for the parametric modeling of flow in different types of hydraulic valves is developed by Jose M. Rodriguez. [2] This methodologies based on derivation of a generalized parametric function for modeling of the discharge coefficient of valve restrictions. This function has a square root form and two parameters that are characteristic of the restriction geometry and that can be derived from numerical CFD simulations. Once the flow coefficient functions are characterized, the calculation of flow rate is done by means of either a second order equation or a simple iterative procedure, in which input data are fluid properties and main geometrical dimensions. The methodology is demonstrated by applying it into two completely different hydraulic valve systems i.e. a brake master cylinder and HCU valves. The results of flow rate calculated with parametric model. And it is compared with those obtained from CFD simulations on initial design and on designs with same topology but different dimensions obtaining in every case. The coefficients of critical restrictions of a particular valve are used to determine the flow for other valves of similar topology but different dimensions. If the geometry is severely changed, critical restrictions must be identified and the parameters must be calculated again. The flow rate can be calculated using iterative or non-iterative methods.

In order to examine the performance characteristics of the parts of high-pressure, cryogenic ball valves, numerical analyses of the strength and thermal shock were conducted and the seat structure was investigated and tested by Dong-Soo Kim.[3] The conclusions are obtained as the design of the constituent parts of the ball valve, including the body, seat, bonnet, ball and spring, were optimized. In this study, a high-pressure, cryogenic ball valve that can achieve zero leakage was designed.

Thus, CFD analysis of symmetric disc valve has been carried out by G. Tamizharasi [4] and conclusions was drawn are at smaller opening angle the pressure loss is comparatively less. The total pressure variation and intensity of turbulence increase at downstream when opening angle increases. Force concentration is observed for 40° valve opening position. Increased wall shear are observed for 40° and 60° opening positions.

The paper of Ana Pereira[5] shows utility of the CFD numerical simulations as a tool for design and optimization of hydropower performance and flow behavior through hydro mechanical devices or hydraulic structures of intake and outlet types. Experimental tests not always are viable because they are very expensive and it is much more difficult to analyze different scenarios and boundaries. The flow of a real fluid in contact with a boundary implies velocity variations, pressures gradients and shear stress development, from which energy losses result, as important factors to take into account in the concept, design, construction, operation and maintenance of hydropower plants or any other type of hydraulic conveyance system.

The results of CFX simulation generated by Xue guan Song [6] agreed with the experimental data very well. However, at some peculiar position, especially at the valve opening degree smaller than 20°, it didn’t agree well. This may be due to disadvantage of the k-ε turbulent model of its own. It’s suggested to use another turbulent model which is good at treatment of near-wall such as k-ω model and SST turbulent model. The simulation by CFX was very sensitive to the degree of valve opening near to fully closed condition, where the flow near the valve is highly turbulent. So small subdivision is recommended near this region, and results are used with the comparison of the test values.

In general, the result obtained by using commercial code ANSYS CFX 10.0 agrees with the experimental result very well. However, it is recognized that all CFD based predictions are never possible to be 100% reliable. Hence further investigation must be performed before the computational simulations are used directly in the industry.

### III. CFD Analysis and theory Approach

CFD provides numerical approximation to the equations that govern fluid motion. Application of the CFD to analyze a fluid problem requires certain steps. First, the mathematical equations describing fluid flow are written. CFX is a commercial Computational Fluid Dynamics (CFD) program, used to simulate fluid flow in a variety of applications. The ANSYS CFX product allows engineers to test systems in a virtual environment. The scalable program is applied to simulation of water flowing past ship hulls, gas turbine engines (including compressors, combustion chamber, turbines and afterburners), aircraft aerodynamics, pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners etc.

This is the initial step in analysis process. The primary purpose of geometry creation is to generate a solid that defines region for fluid flow. This section describes creation of geometry. Dimensions and geometry details of existing model are collected. Model is created using...
CATIA V5R20 software and exported in IGES format. The model of valve shape in shown in the following figures the step defines creation of regions and geometry. 2D region is created for defining inlet and outlet. Creation of regions facilitates to assign boundary condition for inlet, outlet and other defined regions. The model of Ball valve is shown in fig. 1 and its exploded view as shown in fig. 2.

Figure 1. Typical Ball Valve model in Catia V5

Figure 2. Ball Valve with Exploded view

Flow coefficient ($C_V$)

The flow coefficient is used to relate to the pressure loss of a valve to discharge of the valve at a given valve opening angle. Today, $C_V$ is the most widely used value for valve size and pipe system. By using the $C_V$, a proper valve size can be accurately determined for most applications. The most common form used by valve industry is Equation (1):

$$C_{V, ISA} = \frac{Q_{gpm}}{\sqrt{\Delta P_{ISA}} / S_g}$$

Where the pressure drop $\Delta P_{ISA}$ can be measured from static wall taps located 2 pipe diameters upstream and pipe diameters downstream of the valve. $\Delta P$ is the pressure drop in units of psi, $Q_{gpm}$ is in units of gpm, and $S_g$ is the specific gravity of the fluid (1 for water).

Flow Pattern

The fluid, which was modeled as water, is given a uniform velocity of 3m/s at the inlet and zero reference pressure at the outlet. Through rough calculation, the range of Reynolds Number of flow in this study is larger than 105, hence the effect of the Reynolds Number is so small that it can be neglected [5].

Numerical Method

Incompressible and viscous fluid (water) flows through the Ball valve. The flow pattern reveals that flow studied is turbulence flow. To deal with turbulence modeling, the eldest approach, Reynolds-averaged Navier-Stokes Equations (RANS), is utilized. Its common form can be written as Equation (2)

$$\frac{\partial \bar{u}_j}{\partial x_j} = g_j - \frac{1}{\rho} \frac{\partial P}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \bar{u}_j}{\partial x_j} \right)$$

Dynamic flow coefficient ($C_T$)

Hydrodynamic torque $T(\alpha)$ is valve shaft produced by the flow passing through valve at a given valve opening angle $\alpha$. The hydrodynamic torque coefficient $C_T$ is a factor, which is independent of the size of valve. For a given valve and valve opening, it is easy to calculate hydrodynamic flow torque by using $C_T$ times the different pressure drop, Equation (3) shows the relation between $C_T$, $T$, pressure drop and valve diameter.

$$C_T = \frac{T(\alpha)}{\Delta P_{net} \cdot d^2}$$

IV. Visual Inspection Techniques

This is one of the best performance prediction techniques for Ball valve. The valve material is CPVC and as per the ASTM D:2846 for valve material sustainable pressure ratings are passing water at 820°C with 36 kg/cm² pressure for six minute and water at 820°C with pressure of 26 kg/cm² for two hours.

The valve performance is visually inspected as shown in fig. 3. The visual inspection
shows that the valve meets the standard criteria with large amplitude. That is the valve passes water of temperature $82^\circ$ C at pressure of 36 kg/cm$^2$ for 6 minutes and 26 kg/cm$^2$ for four hours satisfactorily.

![Figure 3. Test setup for testing of ball valve](image)

**V. Results and discussions**

In this section experimental results for the performance testing of ball valve are discussed. The experimental setup for the testing of ball valve is as shown in figure.3 in which the inlet water is allowed to pass through ball valve at different pressure from one side and other side is blocked. Then the test results obtained are as shown in figures below;

![Figure 4. Ball Valve leaked from its spindle](image)

![Figure 5. Test result at 36 Kg/cm$^2$ for 6 min.](image)
The fig.5 & fig. 6 shows the result at two different pressures and periods in which as the pressure drop occurs during the testing and also visualizing the water droplets on ball valve then it is said that the piece failed in testing and it is rejected. And if there is no any pressure loss during the testing then it is passes the test and that ball valve piece is accepted.

VI. Conclusion
In this paper, basics of CFD and experimental testing of ball valve has been discussed. Visual inspection shows the valve meets standard criteria as per ASTM D2846 for valves, so the design of valve is tested and validated. It is observed that CFD software method gives similar results matching to visual inspection. As per experimental testing following conclusions about causes for leakage problem are observed; i) Owing to shrinkage in material of spindle, ball or main body of ball valve during moulding process, ii) Owing to improper size and dimensions of small ‘O’ rings which are placed on spindle of ball valve, iii) As there is small bending in spindle during manufacturing then it also causes the leakage and iv) As the fitting of ball valve nut is improper, it may also put the clearance in ball valve which causes the leakage problem.

Acknowledgment: Authors are thankful to the SSBT’s, College of Engineering and Technology, Bambhori, Jalgaon for providing library facility and supreme Industries, Jalgaon. The authors would like to thank the staff and colleagues for useful discussions.

References