Investigation of Ball Valve Design for Performance Enhancement

Vishal A. Andhale¹, Dr. D. S. Deshmukh²

¹Research Student in mechanical engineering department SSBT’s, COET, Bambhori, Jalgaon (MS), India
²Professor and Head, Department of Mechanical engineering, SSBT’s, COET, Bambhori, Jalgaon (MS), India

¹vish8501@gmail.com
²deshmukh.dheeraj@gmail.com

Abstract- Flow is controlled using ball valve is most common process in hydraulic system. As Chlorinated Polyvinyl Chloride (CPVC) ball valve is considered in this paper for study of each coponents. Owing to reliability factor and operating conditions, that is quick opening and closing at high temperature and pressure conditions ball valves are mostly used as flow controlling device. Also generally CPVC ball valves are in use when operating pressure is relatively low. Further scope is observed in designing the valve for leak proof and trouble free performance in hydraulic systems. Fast progress of the flow visualization and numerical technique should be incorporated for further flow visualization through the valve and to evaluate the performance of valve. In this experimental work flow evaluation through Ball valve is accomplished using Computational Fluid Dynamics (CFD) software. CATIA V5 R20 software is used for Modelling of valve and ANSYS FLUENT 14.5 is used for analysis. Amount of water leakage is calculated from numerical analysis and it is compared with experimental data. Difference between experimental results and computational results obtained for two cases shows good agreement with each other.

Keywords-C PVC Ball valve, optimum performance, ICEM CFD14.5, ANSYS FLUENT 14.5, CATIA V5 R20

I. INTRODUCTION

A ball valve is a form of quarter-turn valve which uses a hollow, perforated and pivoting ball (called a “floating ball”) to control flow through it. It is open when the ball hole is in line with flow and closed when it is pivoted 90° by the valve handle. The handle lies at in alignment with the flow when open and is perpendicular to it when closed which makes easy for visual confirmation of the valve's status. It has simple mechanical meeting and small flow resistance in a totally open function. It provides a surprisingly excessive flow capacity. Usually, the fluid friction coefficient is low and additionally the increase is normally minimal because the Ball valve is operated with a quarter turn. Manual operation may be through lever or tools provided with it. They're most suited for exceptionally low pressure with the flow.

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 behaviour 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. The ball has a hole or port through the middle, so that when the port is in line with both ends of the valve, the flow will occur. When the valve is closed, the hole is perpendicular to the ends of the valve and flow is blocked.
Features of CPVC Ball Valve are:

- Durable & Performing well after many cycles.
- Reliable and closing securely even after long periods of disuse.
- Light weight and quick assembly.
- Smaller in size than most other valves.
- Tight sealing with low torque.
- Fire resistance.
- Low Fluid friction coefficient.

II. LITERATURE REVIEW

[1] 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 cavitation 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 cavitation model for valve flow prediction. The significant difference between cavitation and non-cavitations conditions shows importance by taken into account cavitation effects in prediction of the flow rate.

Billy D. Black, David F. Menicucci, and John Harrison [2] studied Analysis of Chlorinated Polyvinyl Chloride Pipe Burst Problems This report documents the investigation regarding the failure of CPVC piping that was used to connect a solar hot water system to standard plumbing in a home. Details of the failure are described along with numerous pictures and diagrams. A potential failure mechanism is described and recommendations are outlined to prevent such a failure. They concluded that The CPVC pipe failure occurred on the cold water inlet pipe leading to the standard domestic hot water heater. The solar heating loop was connected directly to the inlet and outlet connections of the standard hot water heater using Tee fittings. The thermal expansion of the hot water caused it to expand and back up into the inlet pipe since the house side of the plumbing is a closed system and there was no expansion tank. The hot water exceeded the temperature and pressure limits of the CPVC pipe and the pipe bulged and burst.

Arun Azad studied [3] Flow Analysis of Buttery Valve Using CFD. In this research work flow analysis through buttery valve with aspect ratio 1/3 has been performed using computational software. For modeling the valve ICEM CFD 12 has been used. Valve characteristics such as flow coefficient and head loss coefficient has been determined using CFX 12 for different valve opening angle as 30°, 60°, 75°, and 90° (taking 90°as full opening of the valve) for incompressible fluid. Value of head loss coefficient obtained from numerical analysis has been compared with the experimental results.

G. Tamizharasi and S. Kathiresan [4] worked on CFD Analysis of a buttery valve in a compressible fluid. In this paper, three-dimensional numerical simulations by commercial code CFX were conducted to observe the flow patterns around buttery valves with various opening degrees and uniform incoming velocity were used in a piping system. Performance of valves under various operating conditions is generally obtained through an experimental testing of prototype or scaled valves. The availability of performance parameters for compressible flow is limited, and experimental testing can be cost prohibitive. In this case the computational fluid Dynamics analysis provides better results. The capability of using computational fluid dynamics is a test to determine its viability for determining its performance parameters. The objective of the project is to analyze the flow characteristics and performance of buttery valve with the disc shape namely a) Symmetric disc. The analysis would be carried at various valve opening positions (20 deg, 40 deg, and 60 deg) by using CFX tool. A comparative study would be made with the parameters such as static and total pressure, intensity of turbulence, force.

A methodology for the parametric modelling of flow indifferent types of hydraulic valves are developed by Jose M. Rodriguez, [5] 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’s 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 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[6]. 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.

José R. Valdés [7] developed A methodology for the parametric modelling of the flow coefficients and flow rate in hydraulic valves. It is based on the derivation, from CFD simulations, of the flow coefficient of the critical restrictions as a function of the Reynolds number, using a generalized square root function with two parameters. The methodology is then demonstrated by applying it to two completely different hydraulic systems: a brake master cylinder and an ABS valve. This type of parametric valve models facilitates their implementation in dynamic simulation models of complex hydraulic systems.

Thus CFD analysis of symmetric disc valve has been carried out by G. Tamizharasi [8] 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 40⁰ valve opening position. Increased wall shear are observed for 40⁰ and 60⁰ opening positions.

V. J. Sonawane [9] designed and analyzed the Globe Valve as Control Valve Using CFD Software. This paper presents the modeling and simulation of the globe valves. The flow system with globe valves is complex structure and has non-linear characteristics, because the construction and the hydraulic phenomena are associated of globe valves. In this paper, three-dimensional CFD simulations were conducted to observe the flow patterns and to measure valve flow coefficient when globe valve with different flow rate and constant pressure drop across the valve were used in a valve system. Furthermore, the results of the three-dimensional analysis can be used in the design of low noise and high efficiency valve for industry.

H. Kursat Celik [10] studied Determination of flow parameters through CFD analysis for Agricultural Irrigation Equipment. In this study, a sample plastic mini valve has been utilized for CFD analysis. Flow behavior of the fluid in the valve was simulated three dimensionally using a commercial CFD code. Pressure loss, head loss, flow coefficient, resistance coefficient and cavitation index parameters were calculated for different flow rates with different valve opening positions with the aid of simulation results.

According to results of CFD, the pressure loss changes with inlet flow volume rate and valve opening positions. If valve opening angle and inlet flow volume rates re increased, pressure loss, head loss and resistance coefficient are increased but flow coefficient is decreased. Also leakage due to high value sudden pressures, manufacturing or material errors is another problem for this type of plastic mini valve. It may therefore be necessary to evaluate structural deformation cases for optimum part thickness of valves to avoid leakage during its operation.

The paper of Ana Pereira [11] shows utility of the CFD numerical simulations as a tool for design and optimization of hydropower performance and flow behaviour 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 analyse 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.

Results of CFX simulation generated by Xue guan Song [12] agreed with the experimental data very well. However, at some peculiar position, especially at the valve opening degree smaller than 20⁰, 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.

[2] 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.
Vishal Andhale & Dr. D. S. Deshmukh [13] worked on Flow Evaluation of Ball Valve for Performance Enhancement Using CFD Software. 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.

III. EXPERIMENTATION

Hydro Pressure testing machine is used for the experimentation. Visual inspection method is most preferred performance analysis technique 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 82°C with 26 kg/cm² pressure for four hours and water at 82°C with pressure of 36 kg/cm² for six minute. Water is taken as a fluid and it is assumed that it is incompressible.

The performance of valve is inspected visually. It shows that the valve meets the standard criteria with large amplitude. That is the valve passes water of temperature 82°C at pressure of 26 kg/cm² for four hours and 36 kg/cm² for 6 minute satisfactorily.

During the testing; leaked water from spindle of ball valve is collected in flask to measure the amount of discharge. From the sample of ball valve taken for experiment, the following results are obtained which as shown in table 1;

<table>
<thead>
<tr>
<th>Case</th>
<th>Inlet Pressure (Kg/cm²)</th>
<th>Time (Min)</th>
<th>Outlet Discharge (m³/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>36</td>
<td>6</td>
<td>6.75 x 10⁻⁷</td>
</tr>
<tr>
<td>II</td>
<td>26</td>
<td>240</td>
<td>5.27 x 10⁻⁷</td>
</tr>
</tbody>
</table>

IV. CFD ANALYSIS AND THEORETICAL APPROACH

Computational Fluid Dynamics (CFD) has been used extensively to successfully model fluid flow in number of fields, such as aerospace and pump design. It has not been used as much to model the very complex flow through valves. In this study, however, a high end CFD tool was used to numerically predict the point of incipient cavitation in several complex valve configurations. CFD provides numerical approximation to the equations that govern fluid motion. Application of the CFD to analyse 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 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 V5 R20 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. 3

A. Boundary conditions
A. In this analysis inlet and outlet boundary conditions are defined and subjected to two different conditions that are;

Case I:
- Inlet pressure is taken as 36Kg/cm²
- Temperature of 82°C
- Flow direction is normal to boundary.
- Other end of ball valve is set as wall.

Case II:
- Inlet pressure is taken as 26Kg/cm²
- Temperature of 82°C
- Flow direction is normal to boundary.
- Other end of ball valve is set as wall.

The domain for the analysis is taken as fluid domain and the fluid is taken as water. Material taken as CPVC

B. Mesh generation and analysis
The 3-D geometry is generated in the following way. Initially a solid model has been created in CATIA V5 R20 environment. CATIA is a parametric solid/surface feature-based modeller which uses Para solid geometric modelling. The model is then imported to FLOWIZARD for mesh generation using the FLUENT engine in a user friendly environment. The fig. 4 shows the meshing of ball valve assembly.

A typical 3-D mesh has about 2.6–3.0 million elements. The mesh quality has been rigorously checked for parameters such as skewers and aspect ratio and a reasonable balance between mesh size and quality has been attempted. Mesh independence for 3-D simulation was ensured by using three different mesh levels comprising of about 2.0 million, 3.0 million and 4.0 million cells. The numerical values obtained for the accepted mesh were found to agree within 3% to the extrapolated value and accordingly the present mesh was deemed to be adequate. Grids in the mid-plane of valve have shown with the cross-sectional mesh. Unstructured volume mesh is created for this assembly. Grid size of the order of 0.674476 million cells was deployed in this case.
Fig. 5 and fig. 6 illustrates the velocity contours for Case I and Case II respectively. In which it shows the behaviour of water flow in the ball valve.

Fig. 7 and fig. 8 shows the static pressure contours for Case I & Case II respectively. These all figures demonstrate the capability of CFX to simulate the complicated flow in 3-D space. They do not show the flow feature, but also provide obvious evidence of prediction’s validity.

V. RESULTS AND DISCUSSIONS

In following figure 9 and figure 10, an experimental and software results are obtained respectively. In which Case I is taken for the input pressure of 26 Kg/cm² and Case II is for input pressure of 36 Kg/cm².

Figure 9 Experimental Result

Figure 10 Software Result

From figure 9, it is seen that the discharge in experimental analysis for Case I is $5.27 \times 10^{-7}$ m³/sec and for Case II is $6.75 \times 10^{-7}$ m³/sec. Fig. 10 shows discharge from CFD simulation i.e. for case I is $4.86 \times 10^{-7}$ m³/sec and for Case II is $6.39 \times 10^{-7}$ m³/sec.

The Validation of the CFD model is done using the experimental data represented in Fig.11. It compares the discharge evolution in the experimentation and discharge evolution in the CFD simulation for both cases (i.e. Case I & Case II).

The values determined by CFD model is close to experimental results. This shows agreement of
results that is observed value and CFD software values for performance testing of ball valve.

![Graph showing discharge comparison between CFD and experimental results](image)

Figure Error! No sequence specified. Relative error on Discharge between CFD and experimental results

Table No 2 Comparison between Experimental and Numerical results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Case I</th>
<th>Case II</th>
<th>Difference between Experimental and software results in %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Output Parameter</td>
<td>6.75 x10^{-7}</td>
<td>6.39 x10^{-7}</td>
<td>3.6 x10^{-8}</td>
</tr>
<tr>
<td>Discharge (m^3/sec)</td>
<td></td>
<td></td>
<td>5.34</td>
</tr>
<tr>
<td>Experimental</td>
<td>5.27 x10^{-7}</td>
<td>4.86 x10^{-7}</td>
<td>4.1x10^{-8}</td>
</tr>
<tr>
<td>Software</td>
<td></td>
<td></td>
<td>7.78</td>
</tr>
</tbody>
</table>

VI. CONCLUSIONS

Experimental observations and software analysis of ball valve is done. Visual inspection shows the valve meets standard criteria set as per ASTM D2846. As per the experimental observations, following are main reasons of trouble in ball valve operating performance.

- It may be due to shrinkage in material of spindle, ball or main body of ball valve during moulding process
- It may be due to improper size and dimensions of small ‘O’ rings which are placed on spindle of ball valve as a packing
- There may be small bending in spindle during manufacturing
- If the complete assembly is improper or if fitting of ball valve nut is not tighten properly; it may increase clearance in ball valve assembly which also cause leakage problem.

Therefore for reducing water leakage problems as remedial measures, following points must be considered:

- Shrinkage allowance should be considered while designing.
- Proper or accurate size of ‘O’ rings should be used that is with less tolerance limit.
- Straightness checking of spindle should be carefully done.
- While assembling, proper fitting of each part should be done or checked.

II. ACKNOWLEDGMENT

Authors are thankful to the SSBT’s, College of Engineering and Technology, Bambhori, Jalgaon for providing library facility and also for providing laboratory facility. Authors would like to thank the staff and colleagues for useful discussions.

REFERENCES


