Computational design optimization of industrial single piece ball valve to enhance the flow performance

Ravichandra Rangappa\textsuperscript{a,d}, BRN Murthy\textsuperscript{b,*}, Avinash Porali\textsuperscript{a,d}, S Rajesha\textsuperscript{a,d}, N Natarajan\textsuperscript{c}

\textsuperscript{a} Department of Mechanical Engineering, Sahyadri College of Engineering & Management, Mangaluru, Karnataka, India 575007
\textsuperscript{b} Department of Mechanical and Manufacturing Engineering, MIT, Manipal, Karnataka, India
\textsuperscript{c} Caliper Engineering Pvt. Ltd., Adyar, Mangaluru, Karnataka, India 575007
\textsuperscript{d} Visvesvaraya Technological University, VTU, Belagavi, Karnataka, India
\textsuperscript{*}Correspondence E-mail: murthy.brn@manipal.edu Phone: +91 9620867499

Abstract. The characteristics of flow through the fluid flow system largely depends on the control valves and their performance. Ball valves are one among the major valves widely used in various industries due to their simple construction and ease of manufacturing. Thus investigating flow characteristics of these valves is most essential to minimize the losses due to friction and cavitation caused within the valve body. The main objective of the current work is to carry out the computational fluid dynamics analysis using Ansys® Fluent® as solver and Solidworks® as 3D modelling tools to investigate the flow patterns through the single piece ball valve to determine the various flow characteristic and there by suggest design optimization for improved flow rate and performance. Various designs of ball valve such as BVD1, BVD2 and BVD3 were tested through CFD simulation. The simulation results reveals that BVD1 and BVD2 are failed in bidirectional flow characteristics. However BVD3 shows the significant improvement in all the flow characteristics.

Key Words: Fluid flow co-efficient, design optimization, single piece ball valve, CFD

1. Introduction
Valves are one of the major component of processing industries like, petrochemical, food processing, beverages, oil & gas, chemical and others where fluid flow is the major phenomenon. Apart from industries, domestic water supply system, LPG supply lines, agricultural irrigation systems, etc. also use various type of valves with an intention of controlling the fluid flow. Failure of valves or decline in efficiency of valves may affect the processing plants to a large extent. Thus, durability, flow characteristics, efficiency and consistency of the flow control valves in such processing industries defines their productivity. The ball valve is one such valve used in major industries to control the fluid flow rate due to its inherent advantages over other type of valves. Ball valves are simple in its structure, and poses very low resistance for fluid flow and with high performance. The ball valve are also surpass other industrial valves in terms of operational safety and pressure losses in the pipelines [1]. Jablonski and Helfer discussed the application of ball valve in a high temperature environments like refineries. They have mentioned few high temperature applications where ball valves are commonly part of the system are hydrocracker feed with 343 °C, catalytic hydrocracker with 510°C, fuel gas and fractionator
bottoms are with 760℃. [2]. Applications involving cryogenic systems also adapt ball valves to control the fluid flow where two-phase flow scenario appears due to sudden evaporation of cryogenic liquid in contact with atmospheric weather [3].

Cavitation is one of the major problem associated with fluid flow systems. Cavitation is caused due to the drop in hydrostatic pressure which will get similar to the vapor pressure of the fluid by high flow velocities. Subsequently, induces small bubbles along with the stream. These bubbles will naturally collapse when they mix with the high pressure stream while they were carried along with the liquid flow. During final stage of bubble collapse, a high pressure peaks are generated inside the bubbles and in their immediate surroundings. These pressure peaks develops a mechanical vibrations, and noise. Inner wall of the valve body suffers surface erosion due to this vibration force. Severe cavitation also influences the fluid flow property and efficiency of any hydraulic valve. Cavitation is also depended on the tensile strength of the fluid medium [4]. Qian et al (2019) have conducted comprehensive review on cavitation in various valves. They have found the increase in cavitation for increased inlet pressure for most of the valves. For smaller flow rates the cavitation occurs near the valve seat. However, as the flow rate increases, the cavitation moves to downstream. The cavitation can be minimized by improving the shape of cross section, adding a pressure reducing device, or optimizing the valve geometrical structure [5]. Liu et al (2020) carried a research to study the effect of inlet pressure on cavitation characteristics, where they have conducted both experimental and numerical investigation. They have proven that the cavitation length increases with increase in inlet pressure, subsequently it has an ill effect on velocity of fluid flow [6].

Identifying the performance of a ball valve which is made of metal is not a simple or easy phenomenon. Internal fluid dynamics are invisible, thus prediction of the ball valve performance is mainly dependent on the pressure drop across the valve and the flowrate measurements [7]. Fu & Ger, (1998) have proposed mathematical model to determine the flowrate and coefficient of flow using pressure difference between upstream and downstream of fluid across a valve. They also validated the same with an experiments [8]. International Electrochemical Commission (ICE) has proposed the international standards on sizing equations and fluid flow under installed conditions in 1998 under Industrial Process Control Valves, which was adapted by Bureau of Indian Standards (BIS) from 2008 onwards. It has clearly stated that test procedures to be followed during testing and installing of valves [9].

Long and Guan (2011) have compared the Coefficient of flow and Resistance Coefficient to understand the reliable parameter out of the two in characterizing the valve. They have identified that the Coefficient of flow value was influenced by the valve position. If the valve is completely open the experimental and mathematical models were given same values. However, for the partially opened valve, it requires a modified mathematical model to obtain the accurate results. They proposed the Resistance Coefficient model is more dependable than the Coefficient of flow [10]. Influence of open and closing of valve on flow characteristics were studied by Chi et al (2016), in which they did focus on effect of opening and closing duration on the flowrate. The pressure and velocity distribution in the upstream and downstream for various opening and closing timing were also investigated to know the flow characteristics [11].

Yang et al., carried a structural analysis using computational techniques and compared DN300 and DN400 standard valves. Deformation, equivalent stress, factor of safety and flow coefficient are the major parameters which were considered in their study. For the test they have followed IEC 60534-2-3 test standards. They noticed that DN300 has lesser deflection compared with the DN400 standards. [12]. Similarly, P Ebenezer et al., (2015) conducted structural analysis applying Finite Element Analysis for an industrial ball valve to study the deflection of stem and ball due to static pressure. In addition, they also carried out the CFD analysis to investigate the turbulent intensity, velocity magnitude and pressure
distribution across the pipeline [13]. Another computational study in which ball valves used in a multiphase flow with fluid-solid interphase has carried by Askari and Falavand Jozaei (2018) to investigate the effect of erosion on valve body and components. A combined continuous fluid phase with discrete phase modeling were used to simulate the fluid-solid multiphase flow [14].

Turbulence intensity of the flow stream is measured in terms of Reynolds’s number where the number exceeds 4000. Thus, Reynolds’s number is also calculated for the flow under various levels of valve aperture opening. Influence of Reynold’s number on flow coefficient and flow resistance was considered by many researchers. Chern & Wang, (2004) has shown that the Reynolds Number has no influence on Loss Coefficient, Chern et al., (2007) in their work proved similar kind of results. They have shown that for the valve aperture opening less than 80%, Reynolds Number has no influence on Coefficient of Flow. Moujaes & Jagan, (2008) presented the results in terms of Coefficient of Flow versus valve aperture opening for various Reynolds Number flow, they also found that the Flow Coefficient values are relatively close to each other irrespective of Reynolds Number. Thus investigation in terms of Reynolds Number seems to be of no interest in the present work.

In most of the general applications, the fluid flow will be in one direction. Thus, the design features of valves are also unidirectional. Installing them in correct flow direction is most important to avoid the damage to the valve and the flow stream. However, there are several applications like vents, drains, and other applications may desire a bidirectional sealing valves. A proper design is required for such valves to accommodate bidirectional flow [17].

2. Objectives of the present work

The objective of the current study is to optimize the design features for an industrial single piece ball valve for improved flow characteristics and to develop various characteristic curve for the fluid flow while the valve is operating at various aperture openings. Present analysis and optimization is carried with numerical investigation applying Computational Fluid Dynamics focusing on flow characteristics without considering structural analysis. The maximum pressure assigned during analysis was limited to 700 kPa and the effect of valve opening/closing duration was not considered as a factor of study. The performance properties like pressure drop, flow velocity, mass flowrate and flow coefficient are the considered to investigate and optimize the valve design. The fluid flowing through the valve is assumed as water throughout the investigation and no other fluids were involved in the current study.

3. Methodology

For most of the flow control valves, major performance properties assessed are pressure drop, flow velocity, mass flowrate and flow coefficient. Ball valve considered in the present work is developed and manufactured in Caliper Engineering and Lab Pvt. Ltd (Caliper Company) with a simple design and manufacturing features that can ease the production process. The company have adopted a complete machining operation without considering casting to avoid the cost involved in design and development of casting die, pattern core and other accessories. Internal wall roughness cannot be controlled in casted valves; however, machined valves can have improved surface finish. In case of casted valves, for any small variations in the design would incur the additional investment cost. When the vale is manufactured completely by machining, the production of the valve is cost effective. In addition it can be versatile to cater various sizes and flow characteristics as per the customer needs. The exploded view of the valve developed by the Caliper Company and part list of the single piece ball valve considered in this study is shown in Fig 1.
As mentioned in the exploded view (Fig 1) of single piece ball valve, single piece refers to the Hex-Body of the valve, which is machined out of a hexagonal solid/hollow block. High precision CNC Machines and workstations were involved in obtaining the dimensional accuracy. One end of the body is drilled with the hole of flow diameter and the other end of the body is with larger diameter to facilitate the assembly of ball seat, ball, adapter and gaskets. After placing all the necessary parts into the valve body, adapter with an external thread is assembled into the valve body with leak-proof. Fig 2 illustrates the dismantled parts of the ball valve. Caliper Company has designed a hexagonal hole in the adapter body to support an Alan Key for the ease of assembly. Based on the assembly technique, flow pattern of the ball valve are classified in to two designs as shown in Ball Valve Design 1 (BVD_1) and Ball Valve Design 2 (BVD_2) referring to Fig 3. Current study has proposed an improved flow design which is referred as Ball Valve Design 3 (BVD_3) in this study. The valve has been tested and approved by Caliper Company under Shell Test with the static pressure of 70 kg/cm². However, the analysis of performance under fluid flow for the valve while it has been operated in various aperture levels need to be investigated. Which is considered in the present work. The pressure, velocity and flow coefficient of the flow stream needs to be measured while valve is operated at various levels of aperture opening so as to understand the flow characterization.
4. Numerical Simulation

In the current era, computational techniques have made the industrial design and analysis a time and cost effective by avoiding expensive real-time prototype testing. Computational fluid dynamics (CFD) is one of the widely used method among the industrial design engineers and researchers in the applications involving thermos-fluid dynamics to analyze the systems and components to understand their performance and efficiency. Thus, a CFD analysis was considered using Ansys® Fluent® as solver where, 3D modeling of valve and the flow stream were created using Solidworks® solid modeling tool. Ansys® meshing tool was used to generate the grid with hybrid grid feature. Fig 4 illustrates the inlet, outlet and position of valve in the flow stream and Fig 5 demonstrates the mesh for various valve positions. To normalize the fully developed flow condition simulation pipeline was extended six times the hydraulic diameter at the inlet and ten times the hydraulic diameter at the exit. Similar arrangements were observed in the simulation work carried by Cui et al. (2017). The boundary conditions applied in the simulation are pressure inlet, pressure outlet and wall which confines the fluid domain. To reduce the computation burden, half model of the entire computation domain is considered with symmetry boundary condition along the axis of fluid flow.
5. Grid Sensitivity Test

A grid sensitivity test is most essential in computational simulations to avoid the errors and inaccuracies of the results due to domain and computational dependency. Various patterns and applications of grid sensitivity test were studied to understand the process and phenomenon [18] [16] [15]. However, the grid sensitivity test similar to that of Liu et al., (2020) was most opted and thus adopted for the analysis carried in the present work[6]. In the present study, inlet pressure was maintained with 70 MPa for all the cases and elemental edge length were varied from 0.009 m to 0.0007 m. Pressure Outlet (MPa) is the variable parameter under observation to study the consistency of the result based on the grid size. A pressure profile for various grid sizes is presented in the Fig 6, any grid with elemental edge length less than 0.001 m has the consistent pressure outlet. Thus the elemental length for the grid were maintained less than 0.001 m in all the cases of analysis.

| Edge Length Number | Edge Length (m) | Pressure at Outlet (Mpa) |
|--------------------|----------------|-------------------------|
| 1                  | 0.009          | 25.7                    |
| 2                  | 0.008          | 28.0                    |
| 3                  | 0.007          | 29.7                    |
| 4                  | 0.006          | 31.3                    |
| 5                  | 0.005          | 33.0                    |
| 6                  | 0.004          | 33.7                    |
| 7                  | 0.003          | 35.0                    |
| 8                  | 0.002          | 37.3                    |
| 9                  | 0.001          | 42.8                    |
| 10                 | 0.0009         | 42.8                    |
| 11                 | 0.0008         | 42.9                    |
| 12                 | 0.0007         | 42.9                    |

6. Flow Analysis

CFD analysis was carried using Ansys® Fluent® as solver with standard k-epsilon model, a common model used for turbulent flow analysis. Water with density 998.2 kg/m3 and viscosity 0.001003 kg/m-s was used as working fluid flowing through the valve. Solution method used was Pressure-Velocity coupled and SIMPLEC scheme. Pressure, momentum and turbulence equations were solved with second order to achieve higher accuracy. The grid size was maintained with approximately 145000 nodes with minimum elemental length of 0.001 m and number of elements varying from 680000 to 750000 for all the simulations. A constant inlet pressure of 70 MPa was applied at inlet zone. Each valve was tested for forward and reverse flow, where fluid enters from adapter side is considered as forward...
flow and fluid enters from opposite side of the adopter is considered as reverse flow. A total of 24 sets of simulation comprising three valve designs as shown in Fig 3 with forward and reverse flow and operating at 22.5°, 45.0°, 67.5° and 90.0° valve position as shown in Fig 4 were considered. The flow capacity and sizing equations used in the analysis were based on the international standards IEC-60534-2-1:1998 Industrial-process control valves. Since the working fluid is assumed to be incompressible fluid, thus change in volume and density with respect to change in pressure is zero. The equations used during analysis are:

For incompressible working fluid

\[ \frac{\partial \phi}{\partial p} = 0 \]  

(1)

\[ \frac{\partial \rho}{\partial p} = 0 \]  

(2)

Flow Coefficient (\(C_v\))

\[ C_v = \frac{Q}{N_1 \sqrt{\frac{g}{\rho}}} \]  

(3)

In the equation (3), \(G\) is specific gravity of water and \(\partial p\) is change in pressure in kPa, \(Q\) is volumetric flow in m\(^3\)/h and \(N_1 = 8.65 \times 10^{-2}\) is a constant used based on the unit system[9][18]. Pressure and volumetric flowrate at the outlet boundary condition were measured as area average.

7. Results and Discussion

Two of the ball valve designs (BVD1 and BVD2) considered in the present work are with the adapter in hexagonal shape that has created a neck in hydraulic diameter and imposed restrictions for the flow. The forward and reverse flow in such designs would result in different flow characteristics. To investigate the effect of the restriction, analysis were carried both in forward and reverse flow directions for all the valve designs. Valves with bidirectional flow ability have a special interest in the industries. Table 1 illustrates the various flow characteristics of the three valve designs considered in this study with both forward and reverse flow. Pressure outlet, Volumetric Flow, Fluid Velocity and Coefficient of Flow are the major characteristics where measured to analyze the performance of the valve which are illustrated in bar charts as shown in Fig 7.

Pressure at outlet for various adapter design is illustrated in the Fig 7(a), pressure gradually increases with angle of operation which is natural. However, higher pressure can be noticed with BVD3 in comparison with BVD1 and BVD2. This is due to the improved flow characteristics. Volumetric Flow and Flow velocity shown in Fig 7(b) and (d) are the evidence for the improvement of flow in the BVD3. Significant feature noticed in this study is the difference in forward flow and reverse flow for any given design of valve. Under BVD1 and BVD2, except at 22.5° in all other valve opening angles, forward flow has shown improved flow characteristics than the reverse flow. However, BVD3 has shown similar performance under both forward and reverse flow, which implies the BVD design is capable for bidirectional flow with other design criteria stated by M E Beasley (2013). The flow curves shown in Fig 8 illustrates the comparison of valve performance during forward and reverse flow. It shows that, the volume flow rate and coefficient of flow is much better in BVD3 especially at 90° valve opening level.
Table 1. Flow Characteristics of the three Valve Designs Considered under this Study

| Valve Design | Valve Opening Angle (deg) | Pressure (Pa) | Volumetric Flow Rate (m³/s) | Velocity (m/s) | Flow Coefficient |
|--------------|---------------------------|---------------|----------------------------|----------------|-----------------|
|              |                           | Inlet         | Outlet                     | Drop           |                 |
| BVD_1_F      | 22.5                      | 7000000.0     | 9082.1                     | 6990917.9      | 3.85            |
|              |                           |               | 6823839.0                  | 47.88          | 52.56           |
|              |                           |               | 3552554.3                  | 55.24          | 82.79           |
| BVD_1_R      | 45.0                      | 7000000.0     | 1384836.4                  | 6823839.0      | 17.06           |
|              |                           |               | 5693315.0                  | 48.02          | 52.56           |
|              |                           |               | 5615163.6                  | 75.24          | 82.79           |
| BVD_2_F      | 22.5                      | 7000000.0     | 8401.0                     | 6991599.0      | 3.71            |
|              |                           |               | 6846002.3                  | 48.02          | 52.56           |
|              |                           |               | 3496323.6                  | 75.96          | 83.36           |
| BVD_2_R      | 45.0                      | 7000000.0     | 1395642.7                  | 6846973.0      | 15.95           |
|              |                           |               | 5258035.5                  | 46.44          | 51.07           |
|              |                           |               | 5604357.3                  | 73.08          | 80.29           |
| BVD_3_F      | 22.5                      | 7000000.0     | 11890.1                    | 6988109.9      | 3.60            |
|              |                           |               | 6759550.6                  | 19.44          | 21.71           |
|              |                           |               | 3751320.5                  | 73.08          | 80.27           |
| BVD_3_R      | 45.0                      | 7000000.0     | 1741964.5                  | 6759550.6      | 15.88           |
|              |                           |               | 5258035.5                  | 46.08          | 50.96           |
|              |                           |               | 3248679.5                  | 73.08          | 80.27           |

A contours of Turbulent Kinetic Energy is plotted in the Fig 9 for all the three designs while valve opened at 45°. The maximum turbulent kinetic energy was high in BVD3 and low in BVD1. It is due to control of flow by the neck formed in the adapter. The adapter neck has reduced the pressure entering the ball valve after the necking, thus the intensity of turbulent kinetic energy decreased as the dynamic pressure decreased. Fig 10 can be referred in comparison with Fig 9 for each design to verify the pressure in connection with the turbulent kinetic viscosity.
Fig 7. Bar Chart Showing Flow Characteristics for all the Design at every angle of Operation (a) Pressure at Outlet (b) Volumetric Flow (c) Coefficient of Flow (d) Flow Velocity

Fig 8. Graph Representing the Improvement in (a) Coefficient of Flow (b) Volumetric Flowrate comparing BVD1 & BVD2 with BVD3
Fig 9. Contours of Turbulent Kinetic Energy at 45° Valve Position (a) BVD1 (b) BVD2 and (c) BVD3
8. Results and Discussion

Three various designs of industrial ball valve were tested through CFD simulation to investigate the flow characteristics and to identify the ability of the designs for bidirectional flow ability. BVD1 and BVD2 failed in bidirectional flow characteristics and their forward flow was better than the reverse flow. However, proposed improvement in design as depicted in BVD3 has shown a significant improvement in all the flow characteristics. Compared to BVD1 and BVD2 under full-fledged flow around 30% improvement is achieved in BVD3. Further to this, BVD3 is capable of bidirectional with same flow characteristics on either direction of flow. Thus, BVD3 has proven to be potential for industrial applications involving bidirectional fluid flow.

References

[1] C. K. Kim, S. M. Lee, and C. M. Jang, 2019 “Performance analysis of a ball valve used for gas pipelines by introducing nondimensional parameters,” Adv. Mech. Eng., vol. 11, no. 1, pp. 1–10.

[2] J. Jablonski and W. Helfer, 2019 “Using ball valves in high-temperature applications,” Hydrocarbon Processing, vol. 2019.

[3] S. S. Bindu, B. S. Archana, R. I. Vivek, and R. K. E. Reby, 2017 “Experimental and Computational Investigations on two phase flow through cryogenic ball valve using liquid nitrogen,” Int. J. Sci. Eng. Res. Vol., vol. 8, no. 7, pp. 1656–1662, [Online]. Available:
Experimental and Computational Investigations on two-phase-flow-throughcryogenic-ball-valve-using-liquid-nitrogen.pdf

[4] V. K. Ramakrishnan, 1984 “Cavitation in control valves,” 13th Nat. Conf. Fluid Mech. Fluid Power 84, Proc. (Tiruchirapalli, India Dec. 12-14, 1984), vol. II, Tiruch, pp. 417–422.

[5] J. yuan Qian, Z. xin Gao, C. wei Hou, and Z. jiang Jin, 2019 “A comprehensive review of cavitation in valves: mechanical heart valves and control valves,” Bio-Design Manuf., vol. 2, no. 2, pp. 119–136.

[6] X. Liu et al., 2020 “Influence of inlet pressure on cavitation characteristics in regulating valve,” Eng. Appl. Comput. Fluid Mech., vol. 14, no. 1, pp. 299–310.

[7] M. J. Chern, C. C. Wang, and C. H. Ma, 2007 “Performance test and flow visualization of ball valve,” Exp. Therm. Fluid Sci., vol. 31, no. 6, pp. 505–512.

[8] W. S. Fu and J. S. Ger, 1998 “A concise method for determining a valve flow coefficient of a valve under compressible gas flow,” Exp. Therm. Fluid Sci., vol. 18, no. 4, pp. 307–313.

[9] IEC, “IEC 60534-2-1:1998 Industrial-process control valves - Part 2-1: Flow capacity - Sizing equations for fluid flow under installed conditions,” p. 91, 1998.

[10] C. Long and J. Guan, 2011 “A method for determining valve coefficient and resistance coefficient for predicting gas flowrate,” Exp. Therm. Fluid Sci., vol. 35, no. 6, pp. 1162–1168.

[11] B. Cui, Z. Lin, Z. Zhu, H. Wang, and G. Ma, 2017 “Influence of opening and closing process of ball valve on external performance and internal flow characteristics,” Exp. Therm. Fluid Sci., vol. 80, pp. 193–202.

[12] S. D. Yang, C. K. Kim, S. Y. Jeon, and J. Y. Yoon, 2016 “Numerical performance and safety analyses of a retainer-type ball valve for use in a high-pressure district heating pipeline,” Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng., vol. 231, no. 6, pp. 1249–1260.

[13] P. Ebenezer, U. K. G. D. S, and S. D, “Design and Analysis of Industrial Ball Valve using Computational Fluid Dynamics,” Int. J. Recent Trends Mech. Eng., no. October, 2015, [Online]. Available: https://www.researchgate.net/publication/282958183_Design_and_Analysis_of_Industrial_Ball_Valve_using_Computational_Fluid_Dynamics.

[14] A. Askari and A. Falavand Jozaei, 2018 “Numerical Simulation of the Effect of Valve Opening and Particle Concentration on the Erosion Damage in Ball Valves of Pressure Reducing Station,” J. Comput. Appl. Mech., vol. 0, pp. 1–10.

[15] M. J. Chern and C. C. Wang, 2004 “Control of volumetric flow-rate of ball valve using V-port,” J. Fluids Eng. Trans. ASME, vol. 126, no. 3, pp. 471–481.

[16] S. F. Moujaes and R. Jagan, 2008 “3D CFD predictions and experimental comparisons of pressure drop in a ball valve at different partial openings in turbulent flow,” J. Energy Eng., vol. 134, no. 1, pp. 24–28.

[17] M. E Beasley, 2013 “Bidirectional sealing ball valves in power applications,” no. October, pp. 67–69.

[18] R. Rangappa, S. Rajoo, P. M. Samin, and S. Rajesha, 2020 “Compactness analysis of PCM-based cooling systems for lithium battery-operated vehicles,” Int. J. Energy Environ. Eng., vol. 11, no. 2, pp. 247–264.

[19] S. D. Yang, C. K. Kim, S. Y. Jeon, and J. Y. Yoon, 2017 “Numerical performance and safety analyses of a retainer-type ball valve for use in a high-pressure district heating pipeline,” Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng., vol. 231, no. 6, pp. 1249–1260.