

Optimization of Industrial Valve Design Considering Flow Parameter using CFD

Shailesh Wamanrao Lokhande¹, Dr.A.M.Langde²

P.G. Student, Department of Mechanical Engineering, ACET College, Nagpur, Maharashtra, India¹

Professor, Department of Mechanical Engineering, ACET College, Nagpur, Maharashtra, India²

ABSTRACT: There are so many valves applicable in industry, one of them is ball valve because ball valve is widely used in power plant to control and regulate the flow through hydraulic plant. It is important to design the valve in such way that optimum efficiency can be achieved in plant by considering the flowing parameter such as pressure drop, velocity, and viscosity etc. CFD analysis improve the valve performance and valve life in industry at desire valve closing angle such as 0°, 15°, 30°, 45°, 60° by changing the ball valve shape and checking the pressure drop at certain angle for getting best result.

KEYWORDS:Text Ball valve, CFD, flowing parameter.

I. INTRODUCTION

A valve acts as a control device in a larger system; it can be modelled to regulate the flow of material and energy within a process. There are several different kinds of valves (butterflyball, globe, etc.) selection of which depends on the application and chemical process in consideration. The sizing of valves depends on the fluids processing unit (heat exchanger, pump etc.) which is in series with the valve. Valves need to be modelled to perform effectively with respect to the process requirements. Efficient modelling of the valves can optimize the performance and stability of a process as well as reduce development time and cost for valve manufacturers.

Valves are commonly used in industrial applications to control the internal flow of both compressible and incompressible fluids. Pressure loss, flow coefficient, loss coefficient, torque coefficient etc. performance coefficient including in valve. Valve generally use both way in circuit for maintained flow .it give in the form of laminar .helps to reduced pressure loss.

Performance factors common to a valve include the following: pressure drop, hydrodynamic torque, Flow coefficient, loss coefficient, and torque coefficient. While these values can usually be obtained experimentally, it is sometimes not feasible or possible to calculate the performance factors of some valves. Another method where in valve performance factors can be obtained is by using Computational Fluid Dynamics (CFD) software to simulate the physics of fluid flow in a piping system around a valve. This study sought to compare experimental and simulated CFD performance factors of a diameter valve for various valve openings and flow conditions in order to determine the validity of using CFD to predict valve performance factors. In this project we are undertaking the Optimization of Industrial Valve design considering the Flow parameters using Computational fluid Dynamic Analysis; this will help us to improve the existing designs of the valves. Now in present day different type of valve use in different application in industries. So many internal problem, flow problem create in valve or around the valve. Such disturbance factor affected on valve .therefore performance of valve gets down. First we will do comparatively study of some other different industrial valve now use in industries. In this project we are undertaking the Optimization of Industrial Valvedesign considering flow parameter using Computational fluid Dynamic Analysis; this will help us to improve the existing designs of the valves.

II. PROBLEM FORMULATION

Valves are commonly used in industrial applications to control the internal flow of both compressible and incompressible fluids. A valve typically consists of a metal disc formed around a central shaft, which acts as its axis of

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

rotation. As the valve's opening angle is increased from 0 degrees (fully closed) to 90 degrees (fully open), fluid is able to more readily flow past the valve. Characterizing a valve's performance factors, such as pressure drop, hydrodynamic torque, flow coefficient, loss coefficient, and torque coefficient, is necessary for fluid system designers to account for system requirements to properly operate the valve and prevent permanent damage from occurring. An attempt to simulate and predict the measured valve performance factors. The Using general purpose CFD software an attempt to simulate and predict the measured valveflow parameter. The Using general purpose CFD software the previously described experiment was modelled in overall approach to simulating the experimental model consisted of using a computer aided design model of the industrial valve with appropriate cylindrical parts added to simulate pipe flow upstream and downstream of the valve, properly meshing and applying physical models to the simulation, creating appropriate flow conditions to match the experiment, such as mass flow rate, fluid properties, valve angle opening, etc., ensuring iterative and grid convergence and time independent results, and recording the predicted flow performance factors and overall flow properties.

III. LITERATURE SURVEY

1. ***A three-dimensional, time-dependent analysis of flow through a bi-leaflet mechanical heart valve: Comparison of experimental and numerical results M.J. King, J. Corden, T. David, J. Fisher***

The flow through a bileaflet mechanical heart valve during the first half of systole was predicted using computational fluid dynamics (CFD). A three-dimensional model of the geometry of the ventricle, valve, sinus and aorta was developed. Flow through the valve was assumed to be Newtonian and laminar. The peak systolic Reynolds number was 1500 based on the aortic radius and the mean aortic velocity. Flow visualisation and laser Doppler anemometry (LDA) experiments were performed and the results were compared to the CFD model. Good agreement between the LDA measurements and CFD predictions was found in the jets through the major orifices of the valve.

2. ***Discharge Coefficients for Incompressible Non-Cavitating Flow through Long Orifices Lichtarowicz, R. K. Duggins, E. Markland***

The results of a number of previous investigations of the discharge characteristics of parallel-bore orifices with length/diameter ratios up to 10 and Reynolds numbers up to about 105 are collected and discussed, together with new data which extend to Reynolds number as low as unity. Simple empirical expressions, which fit the data well, are suggested for design purposes. Biomechanical analysis of hard tissue responses and injuries with finite element fullhumanbody model.

3. ***CFD analysis of the ball valve performance in presence of cavitation. NimaTavakoliShifaji , Gholam Raja AzizyanGholamHosseinAkbari***

In this paper ball valve was analysed computational result and experiment result .the inspection can be predicted using the cavitation index,calculated the pressure data .cavitation create excessive leakage .they directly affected to the valve .cavitation destroy the part within the minute .hence they find the critical condition of valve and designer considering this critical value changes the design and avoid the future problem.

4. ***Design and analysis of globe valve as control valve using CFD software V.J.Sonwane, T.J Rane, A.D Monde,R.V.Vajarinkar,P.C.Gawade***

Recently occurring problem higher valve torque due to the high thrust force acting on the disc, difficult manual operation. High frictional forces at the steam treads and yoke sleeve collarface lead to high torque and reduce valve life. Problem discussed in this paper is about difficulty in manual operation and controlling flow of fluid

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

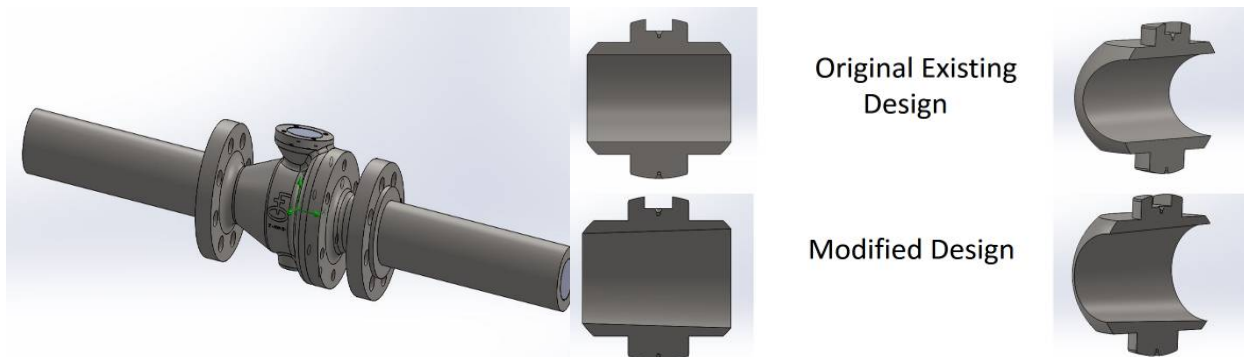
Vol. 5, Issue 6, June 2016

IV. RESEARCH METHODOLOGY

1. Data gathering of the existing industrial ball valve
2. Finite Element Analysis(CFD) of existing industrial valve
3. Optimization of existing industrial valve for process parameters like pressure drop and velocity drop
4. Comparison
5. Recommendations

V. CFD RESULTS

Figures shows the results of CFD analysis of flowing parameter



(a) Pressure drop = Inlet Pressure - Outlet Pressure
Change in velocity = outlet velocity - inlet velocity

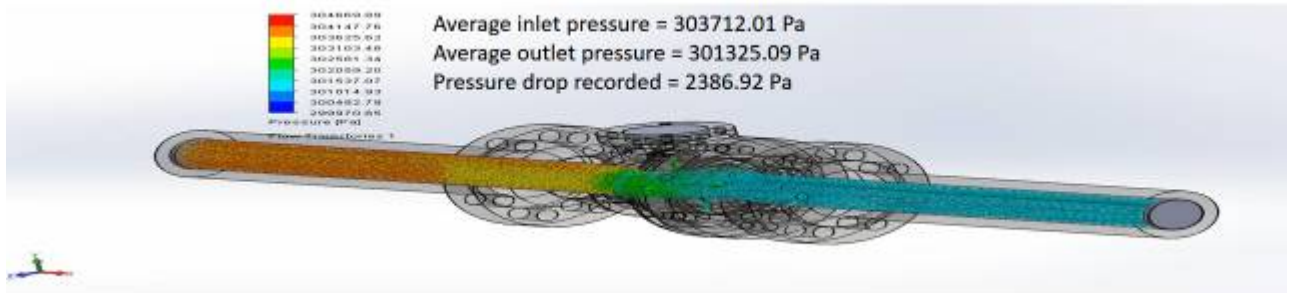
(b) Pressure Drop α Cavitation index

International Journal of Innovative Research in Science, Engineering and Technology

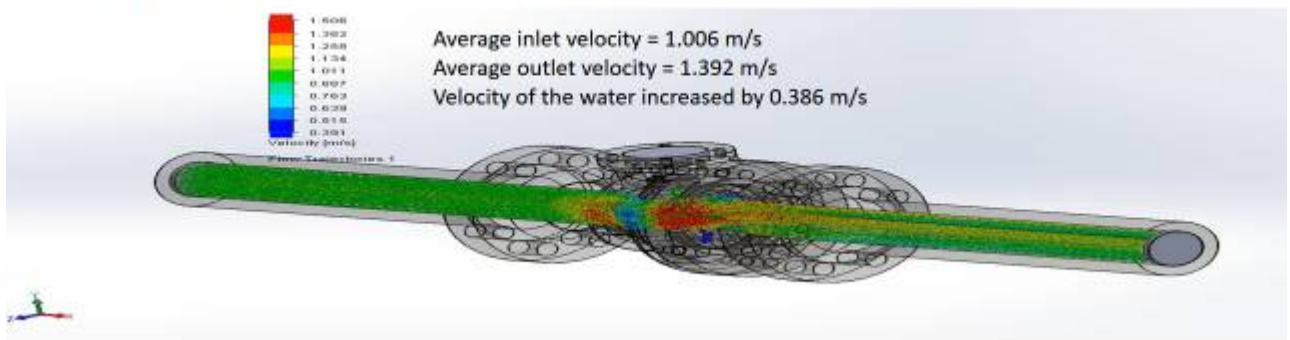
(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

Pressure drop across the valve with valve 30° closed of modified Ball valve



Velocity drop across the valve with valve 30° closed of modified Ball valve



(f) Modified Design

ANGLE	ORIGINAL EXISTING BALL VALVE		MODIFIED BALL VALVE	
	Pressure drop	Change in velocity	Pressure drop	Change in velocity
0°	244.52 Pa	0.005 m/s	213.81 Pa	0.116 m/s
15°	804.92 Pa	0.44 m/s	679.52 Pa	0.122 m/s
30°	2860.84 Pa	0.395 m/s	2386.92 Pa	0.386 m/s
45°	13658.51 Pa	0.559 m/s	10323.04 Pa	0.808 m/s
60°	144063.51 Pa	0.996 m/s	140546.61 Pa	0.864s

VI. CONCLUSION

The modified design was able to achieve the desired results i.e. Pressure drop decreased from 2860.84Pa to 2386.92Pa and Change in velocity decreased from 0.395m/s to 0.369m/s which provide higher efficiency of flowing.

REFERENCES

- [1] "Friction Losses in Pipe Fittings"(PDF), Western Dynamics, LLC., retrieved September 11, 2006.
- [2] "Valve Sizing and Selection." The Chemical Engineers' Resource Page. 1442 Goswick Ridge Road, Midlothian, VA 23114. retrieved Sept 24, 2006.
- [3] M-J., Wang,C-C.,C-H.,Ma, 2007.Performance Test and Flow Visualization of Ball Valve.Experimental Thermal and Fluid Science, Vol. 31: 505-512.



ISSN(Online) : 2319-8753

ISSN(Print) : 2347-6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

- [4] Baldyga, J., Makowska, L., Orciuch, W., Sauter, C., H., P., Schuchmann, B., 2009. Agglomerate Dispersion in Cavitating Flows. *Chemical Engineering Research and Design*, Vol. 87(4): 474-484.
- [5] Mahesh Kamkar, S.R. Basavaraddi, "CONCEPTUAL DESIGN AND ANALYSIS OF HIGH PRESSURE BALL VALVE", *International Research Journal of Engineering and Technology (IRJET)*, Volume 02, Issue 03, pp 1185 -1189, (2015).
- [6] A.S. Tabrizi, M. Asadi, G. Xie, G. Lorenzini, C. Biserni, "COMPUTATIONAL FLUID DYNAMICS BASED ANALYSIS OF A BALL VALVE PERFORMANCE IN THE PRESENCE OF CAVITATION", *Journal of Engineering Thermo physics*, Volume 23, Issue 1, pp 27-38, (2014).
- [7] S. Bagherifard, I. Fernández Pariente, M. Guagliano, "FAILURE ANALYSIS OF A LARGE BALL VALVE FOR PIPE - LINES", *Engineering Failure Analysis*, Volume 32, pp 167-177, (2013).
- [8] Ying Sun and Ting - Rong Ren, "NEW METHOD OF THERMODYNAMIC COMPUTATION FOR A BALL VALVE: COMPUTER SIMULATION OF WORKING PROCESS" Department of power machinery, Xian Jiao - tong university, Xian, Shaanxi province, pp 343 - 353, (1995).