https://doi.org/10.46335/IJIES.2023.8.2.6 **e-ISSN: 2456-3463**
Vol. 8, No. 2, 2023 PP. 26-28

International Journal of Innovations in Engineering and Science, www.ijies.net

A Review on Design and Analysis of Industrial Ball Valve Using Computational Fluid Dynamics

Mr. Harshal Rajesh Dorsatwar [1], Dr. Prashant S Kadu[2]

[1]M.TECH Student, [2]Principal
Abha Gaikwad Patil College of Engineering , Nagpur, India ,441108

dorsatwarharshal@gmail.com

***Received on**: 03 April,2023* ***Revised on**: 27 Aprl,2023* ***Published on**: 29 Aprl,2023*

***Abstract** – Computational Fluid Dynamic analysis is carried out to establish a robust affiliation between the design variables of material design domain and product design domain. The CFD analyses performed for both ball valve and gate valve is necessitated with input parameters that outfits the application such as pressure, density, viscosity and temperature. The maximum pressure acting over diverse regions of the valve system that crop up due to fluid flow was examined by the extension of pressure concentration for different fluids viz. water, lubricant and diesel. The analysis is presumed to be conversant with material selection strategies that satisfy the criterions for the new product development and therefore well defined inputs inclusive of virtual solid model, boundary conditions are promoted with higher grade mesh resolutions. In these cases, approximate selections are exercised and numerical scheme of properties has been adhered to embrace perfection in simulation analysis. The CFD study exemplifies accurate regions wherein maximum pressure assaults the valve body and so the observations originate to ascend product development without the expense of physical testing .Furthermore valve deformation and valve performance is obligatory for material and product design integration and hence customary predictions is done by coupling the CFD results with finite element analysis.*

Keywords- Ball valve, CFD Analysis , Ball Valve Experimental CFD Analysis

I -INTRODUCTION

Valves are components which controls the fluid flow and pressure of a system. Types of applications for valves differ on their own and are normally used on safety and flow control grounds. While these valves are used for flow control, it is obvious that the dynamics of the valve and its control loop has to match strictly with dynamics of the control system. In the course of study on these facets it is over and done with establishment of relationship between valve positions, pressure drop and flow which is commendable as highly non-linear. The able-bodied documentation of valves is not admirable for the reason; these facts features technical hitch to predict the properties down or up-sized. In any flow system the familiar flow restrictors are the valves and hence its design and performance analysis will be a significant task. The selection of the valve types, design and material plays a vital role in its performance and reliability. A number of researches have been experimented in valves for its shape, size, fluid types, operating parameters, discharge coefficient, and eroding characteristics for the improvement of valve technology.
It is quite disturbing that detailed investigation was not done over the integration of material design and valve design to suit a specific functionality. This research work is focused on establishment of relationships between the design variable of both material and product design domains. Computational fluid dynamics a powerful analysis tool is utilized to compute flow restrictions in

https://doi.org/10.46335/IJIES.2023.8.2.6 e-ISSN: 2456-3463
Vol. 8, No. 2, 2023 PP. 26-28

the form of resistance co-efficient and flow volume co-efficient.

II - LITERATURE SURVEY

Industrial ball valve is designed by integral formulations, differential formulations and also computer simulation is Industrial ball valve is designed by integral formulations, differential formulations and also computer simulation is needed for validating every result computational fluid dynamic has been used as research and design tool in the modeling of ball valve. A search on the different simulation models from literature are given as follows:

1) **"Performance test and flow Visualization of ball valve" Ming-JyhChern, Chin-Cheng Wang, Chen- Hsuan ma:**

The performance, flow patterns and cavitations phenomena of a ball valve are examined experimentally. Different patterns of flows in and downstream the ball valves with respect to different valve openings and inlet velocities are envision using a particle tracking flow visualization method (PTFV). Meanwhile, cavitation phenomena are observed under certain conditions. Co-efficient regarding to the performance of valve are determined by pressure and flow rate measurements. The correlations between the valve work and the flow patterns are presented and discussed. The proposed method provides an effective way to determine the performance co-efficient of a valve and to understand the condition for the inception of cavitation.

2) **"Optimization of Industrial Valve Design Considering Flow Parameter using CFD" Shailesh Wamanrao Lokhande, Dr .A.M. Langde**

There are so many valves applicable in industry, one of them is ball valve because ball valve is mostly used in power plant to control and regulate the flow hydraulic plant. It is important to design the valve in such way that best. 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 Degrees by changing the ball valve shape and checking the pressure drop at certain angle for getting best result.

3) **"Design and Analysis of Industrial Ball Valve using Computational Fluid Dynamics" P. Ebenezer Sathish Paul, G. Uthaya Kumar, S. Durairaj, D. Sundarrajan**

Computational Fluid Dynamic examination is carried out to establish a robust affiliation between the design variables of material design domain and product design domain. The CFD analysis performed for both ball valve and gate valve is necessitated with input parameters that out fits the function such as pressure, density, viscosity and temperature. The maximum pressure acting over diverse regions of the valve system that crop up due to fluid flow was examined by the extension of pressure concentration for different fluids viz. water, lubricant and diesel. The analysis is presumed to be familiar with material selection strategies that satisfy the criterions for the new product development and therefore well-defined inputs inclusive of virtual solid model, boundary conditions are promoted with higher grade mesh resolutions.

4) **"Investigation of Ball Valve Design for Performance Enhancement" Vishal A. Andhale, Dr. D. S. Deshmukh**

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 component. 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.

https://doi.org/10.46335/IJIES.2023.8.2.6 **e-ISSN: 2456-3463**
Vol. 8, No. 2, 2023 PP. 26-28

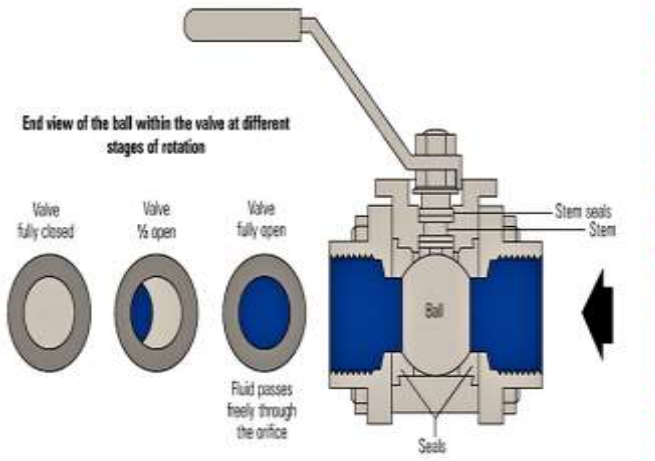
International Journal of Innovations in Engineering and Science, www.ijies.net

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.

III- METHODOLOGY

Computational fluid dynamics (CFD) is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. The various steps involved in solving a flow problem in CFD approach involve the following basic steps.

- Deformation of the Body
- Stress Induced in the Body
- Deformation in the Stem
- Stress Induced in the Stem



IV-CONCLUSION

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is for superior to the manufacturing costs which would accrue if each sample was actually built and tested. From the above structural analysis on valve assembly at different Ball opening position the maximum stress found is 142.65 Mpa in the Ball at 50% opening condition and is below the yield stress 270 Mpa. So, the boundary condition applied in the ball valve is within the limit, the component is safe.

REFERENCES

[1] *"Performance test and flow Visualization of ball valve" Ming-JyhChern, Chin-Cheng Wang, Chen-Hsuan ma*

[2] *"Optimization of Industrial Valve Design Considering Flow Parameter using CFD" Shailesh Wamanrao Lokhande, Dr.A.M.Langde*

[3] *"Determination of Flow Parameters through CFD Analysis for Agricultural Irrigation Equipment: A Case* ***Study for a*** *Mini Valve" H. Kursat CELIK, Allan E. W. RENNIE, Davut KARAYEL, Ibrahim AKINCI*

[4] *"Design and Analysis of Industrial Ball Valve using Computational Fluid Dynamics" P. Ebenezer Sathish Paul, G. Uthaya Kumar, S. Durairaj, D.* ***Sundarrajan***

[5] *"Investigation of Ball Valve Design for Performance Enhancement" Vishal A. Andhale, Dr. D. S. Deshmukh*