Computational Fluid Dynamics (CFD) Analysis Of Butterfly Valve

Lakhan Dangi[1], Mrigendra Singh[2]
[1]Dept of Mechanical Engineering
[2]Assistant Professor, Dept of Mechanical Engineering
[1, 2]LNCT, Indore, M.P

Abstract- *The system proposed in this paper is butterfly valve. The butterfly valve is a type of flow control device, which is widely used to regulate a fluid through a section of pipe. Currently analyses and optimization are of special important in the design and usage of butterfly valves. For the analysis, finite element method (FEM) is often used to predict the safety of valve disc, and computational fluid dynamics (CFD) is commonly used to study the flow characteristics of valve. However it is difficult to obtain the accurate results for the optimization of butterfly valve due to the high non-linearities. For this reason, metamodels and surrogate model methods are employed.*

Keywords- Computational fluid dynamics- CFD finite element method (FEM)

I. INTRODUCTION

Butterfly valve is type of control valve which is used to regulate the flow of fluid though the header pipe. The valve body comprises of central disc which is either nylon coated cast iron or the SS416. The material of disc is selected as per the quality of the fluid. For more accurate fluid flow the SS disc are used. The disc is rotated by either manual handle or the electrical actuator, which is perpendicular to the flow. Despite of any flow the valve can be mounted at any position so that the space is not the constraint for mounting. The beauty of this valve is that the disc rotates only from 0 degrees to 90 degrees. So only quarter turn angular rotation is required. Same size valve can be used for different pressure only valve and disc material of construction is to be changed. Many of the times for varying differential pressure the double disc valves are also incorporated. The seat where the disc is tightly closed also plays a major role. Some manufacturers also provide replaceable seat for easy removal of seat. These valves can be installed in many ways. As a matter of design and for obtaining the proper flow at desired pressure and flow velocity these valves installed in flow to gravity or against the gravity. There are various advantages of installing the valve in gravity flow such as flow vectors of velocity is minimum and the resistance to wall is also minimum. The disc in the valve signifies the flow opening and closing. The general arrangement drawing is shown in figure1. The figure also portrays the various parts of the valve. This includes valve body, valve seat, and stem for disc operation. The valve body is constructed in DIN GGG40, which is of ductile cast iron and having density of 7300kg/m3 and Tensile strength of 390Mpa.

II. SETUP FOR CFD

Computational fluid dynamics can be done by using various analysis software’s. Here we have conducted this analysis by using Ansys CFD. The performance factors which are to be considered for analysis are taken as a input conditions. Before that the 3-D model required for analysis was modelled using solid edge and same was imported to Ansys for further analysis. The modelling was done as parametric model so as to do the meshing properly. Proper meshing will direct to physical simulation of the model by creating accurate flow conditions such as mass flow rate, pressure conditions, valve disc opening angle, Valve disc closing angle. As a system one cannot share between the models of two established systems. CFD is the media to connect the analysis data and setup data. Recording of the data at actual was collected from Belimo site for understanding. CFD is the prediction of input conditions over the boundary conditions by iterations. Iterations depend on the severity of the results required. More the iterations more the accurate results. Analysis is conducted from angle 10 degrees to 90 degrees. The mesh can be done under coarser, moderate and fine refinements. Irrelevant input conditions may lead to uncertain CFD model and particularly for valve inlet and outlet conditions which are to be considered for internal flow. Rather than outlet we are more concern about the inlet conditions. So inlet conditions are of special interest. As mentioned in table 2.1 we will conduct the CFD analysis for given flow conditions. The analysis is conducted on given valve opening angles as given in table. For the same flow simulation start-up boundary conditions are developed and based on these conditions post processing results and investigated and forecast is achieved for various streamline functions. The CFD tool has made a vast development to generate the validated loss coefficients for system simulations.

Whether the pressure drop is less the turbulent flow analysis can be achieved by advanced simulations. The only thing is we need to increase the iterations for analysis. The iterations predict the velocity vectors in ordinate and abscissa with residual scale contours which shows the velocity of the fluid in motion. This turbulent velocity vectors are then removed and necessary vectors are used as boundary conditions for valve simulations. This way of simulating turbulent velocity vectors results in exceptional analytical throughput for inlet boundary conditions. We have used artificial boundary as a 15 volume case instead of pipe dimensions. This saves the incorporation of long pipe and other measuring instruments. Butterfly valve will pass through volume case instead of cylindrical pipe, thus reducing the required for iterations to solve the simulation. By doing this we have reduced the domain size of the valve assembly. Brief representation of the computational fluid dynamics will be discussed in next few articles with Ansys CFD.

III. MESH GENERATION

The goal of meshing is provide the exact solutions for simulations to be conducted in solver. In Ansys we can replace the mesh system with model application. Patch conforming meshing is the type of meshing which derives all part faces and edges conforming all mesh areas. Mainly patch meshing is used for very complex geometries which need to be meshed at each and every node. Patch meshing is invariant of loads and boundary conditions, which gives the parametric meshing as if there any change in part geometry the mesh will update thereon. In numerical analysis the volume mesh represents mathematical description of geometrical entity. This method is termed as topology. If the meshing is not done as per topology meshing will fail with the topology checks. Topology checks ensure the geometry topology associated with mesh topology. In some of the cases topology checks is more stringent than required, it may require to validate the mesh generation and apply conditions to the feature. By solving Navier-Stokes equation using CFD, fluid flow properties like velocity, velocity vector, turbulence and pressure can be simulated and compared with experimental results. Thus CFD simulations and predictions can be used to improvement of real time situations.

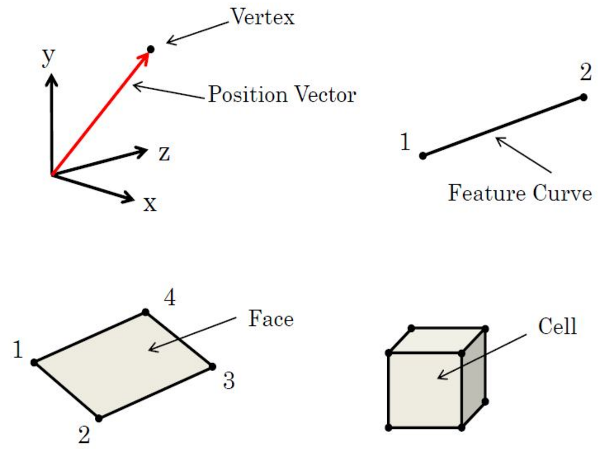


Figure 3.1 Volume mesh components for Individual analysis

3.1 IMPORT OF GEOMETRY

The body diameter of the valve is more the pipe diameter so as to fit the wafer fittings of the valve. Pipe outside diameter matches the valve diameter and inside diameter to the valve seat diameter and hence leakage is arrested from the pipe fittings. Here for analysis we have considered pipe length as 5D, as actual it is maintained as 2.5D, where D is valve diameter. The inlet length of pipe is of much interest as flow simulation will take place before it crosses the valve disc. The flow pattern of the fluid flow in the inlet pipe with inlet fluid velocity and pressure of the fluid before it approaches the outlet flow pipe. After deciding convergence and exact volume flow of valve as shown in table 2.1 the inlet boundary conditions are designed, which will predict the flow analogy. Prediction against the feature is to be carried out is to be placed in virtual enclosure which is also to be meshed for further analysis. Enclosure is surface entity which can be meshed easily as compared to solid entity by which the nodal mesh quality is much more improved and post processing becomes an easy task. Enclosure is a cylindrical surface which covers the solid entity and is used as a boundary condition constraint. Every CFD analysis requires minimum three constraints as inlet, outlet and part feature; hence it is mandatory to mention the inlet and outlet conditions for predictions. One of the faces is treated as inlet and other end as outlet and the part is allowed to pass through between these constraints. Once the mesh is generated and next is to define the inlet boundaries and the constraints for the inlet such as velocity, pressure and turbulence vector. For our study we will not define the turbulence as this will make the analysis easier. While selecting the inlet face for constraint one should be careful for part placement of which the prediction is to be carried out. The inlet conditions can be manipulated as required by changing the parameters of the constraint.

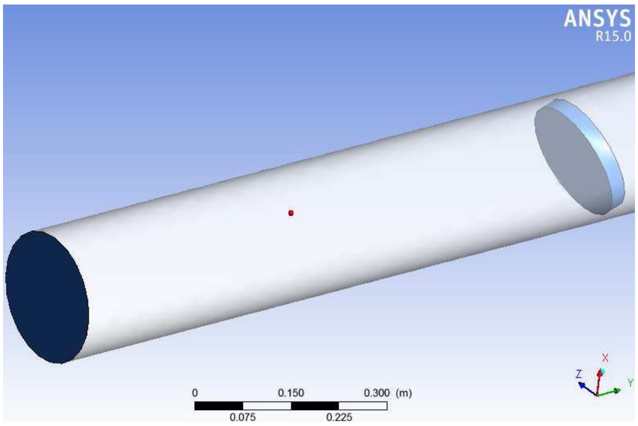


Figure 3.3 Inlet velocity constraints for partially opened valve disc

3.2 BOUNDARY CONDITIONS FOR VALVE DISC

In 3-D entities the boundaries are enclosure type and are surface regions which are enclosed outside the part and define the boundaries by which we can select the required constraint such as wall solid, velocity vectors, pressure gradient, internal surface and flow outlet. Some important boundary conditions we will discuss in below articles.

3.2.1 INPUT AT WALL BOUNDARIES

Wall boundary conditions are used to intact fluid and solid regions present in the region. In viscous flows, there is no-slip boundary condition enforced at walls by default, but by specifying a tangential velocity component in terms of the translational as well as rotational motion of the wall boundary, or models a "slip" wall by specifying shear. The shear stress and heat transfer rate across the fluid and wall are computed based on the flow details in the local flow field.

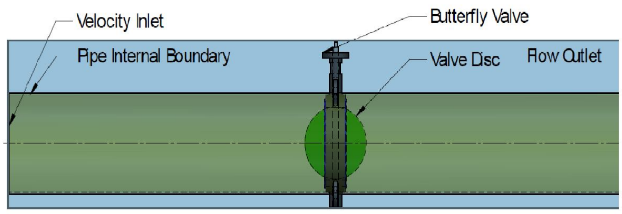


Figure 3.4 Valve and pipe boundaries for mesh simulation

3.2.2 VELOCITY VECTORS

The inlet velocity of the enclosure from which the fluid enters the pipe and change in pressure in convergence takes place along the pipe are termed as velocity vectors. These vectors can be interpreted through the simulation and contour graphics in CFD. Velocity at inlet i.e. the kinetic energy is converted into turbulence energy and is interpreted by k- Epsilon -2 equation with standard wall function should be initiated. Here Prandtl number and turbulence viscosity are ignored as same are not required at this stage of analysis.

3.2.3 FLOW OUTLET

Flow outlet is the total volume flow discharged at outlet after passing through the disc. Valve disc opening can specify the volume at discharge leaving the outlet pipe with velocity gradient and turbulence with zero gradients of outflow. The pipe length of the valve system decides the flow pattern of the system, if the length is not enough to convert the inlet flow to fully developed flow; failing to which the prediction may hamper the results. For effective simulation the pipe length before and after the valve disc is maintained as 5D so that any turbulence effect will not initiate. However we can check the difference between various pipe lengths such as 3D to 8D and record the variance in readings of the flow and validate the flow patterns. The flow pattern in butterfly valve depends on the disc projected area and the angle of opening, upstream and downstream in the valve varies as the flow velocity changes and pressure on the line deviates. Some valve manufacturers design the valve disc in two different fashions as straight plate and curved type. In the straight type the close-off pressure is less than the curved type. The curved type helps in easy removal of fluid which helps to reduce the pressure gradient and hence the velocity recovery is faster and losses are reduced. As the butterfly valve is only quarter turn application, flow attainment after 45 degrees of disc opening is much sluggish.

IV. CALCULATIONS

1. Loss of pressure:-

$$\Delta P = P_{i\theta} - P_{f\theta}$$

2. Torque Calculation:-

$$T_{C\theta} = T_{\theta} + T_{b\theta} + T_{g\theta} + T_{k}$$
$$T_{C\theta} = T_{\theta} - T_{b\theta} - T_{g\theta} - T_{k}$$

3. Coefficient of flow:-

Coefficient of flow for valve Cv is measure of flow rate across the valve at 15.50C at pressure drop of 1 psi or 214 Pa:-

$$C_{v\theta} = Q\frac{\sqrt{SG}}{\Delta P\theta}$$

4. Loss Coefficient:-

Loss coefficient also known as fluid resistance coefficient K□ used in fluid system design, also predicts head loss in the system due to various design inputs. The loss coefficient can be explained as

$$K_\theta = \frac{2gh_{L\theta}}{V^2{}_{avg}}$$

5. Table shows flow data through butterfly valve:-

θ (degrees)	Flow (m^3/h)	Pressure before disc Bar	Pressure after disc Bar
10	178	14.5	14.2
20	400	14.2	13.9
30	650	13.8	13.6
40	1031	13.5	13.1
50	1575	13.1	12.6
60	1981	12.3	12
70	2286	10.2	9.8
80	2550	9.1	8.6
90	2680	8.8	7.6

6. Computational Fluid Dynamics:-

For incompressible flow:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

7. Naïve Strokes Equation:-

$$\rho \frac{D\vec{V}}{Dt} = \rho\vec{g} - \nabla P + \mu \nabla^2 \vec{V}$$

Where,

$$\frac{D}{Dt}(\vec{V}) = \frac{\partial}{\partial t}(\vec{V}) + u\frac{\partial}{\partial x}(\vec{V}) + v\frac{\partial}{\partial y}(\vec{V})$$

$$\nabla^2(\vec{V}) = \frac{\partial^2}{\partial^2 x}(\vec{V}) + \frac{\partial^2}{\partial^2 y}(\vec{V})$$

Also certain assumptions were made for conduct the analysis

- Common assumptions are:-
  Incompressible $\rightarrow$ ρ and μ are onstant
- Others assumptions:-
  Neglect gravity $\rightarrow g = 0$

8. Solution Erros:-

$$E = \frac{f_{exact} - f_{numeric}}{f_{exact}}$$

V. RESULTS

For viewing the results of post-processor contact region has to be defined from which the fluid will come in contact while flowing through pipe. The purpose of the this analysis is to derive the fluid flow at different valve disc opening angles assuming same fluid velocity and differential pressure of 1 bar; however the valve is designed for 12kPa close-off pressure. Close-off pressure is pressure sustained by the valve disc at 0 degrees at full fluid velocity with tight shut-off. The purpose of valve disc is to allow required amount of flow with maintaining the fluid pressure. Fluid flow visualisation is shown for valve disc opening from 10 to 90 degrees for analysis. As Ansys works on mesh, the results are then exported for CFD post and are analysed to lower down the iterations and mesh nodes and elements. Lesser the nodes so as the solution iterations, the optimised solution is to get the post processing results in less iterations. Following are the results of the valve disc operated from 10 degrees to 90 degrees and are explained in detail with plots for velocity vectors and pressure contours by which we will be able to predict the flow through the butterfly valve at different pressure and velocity.

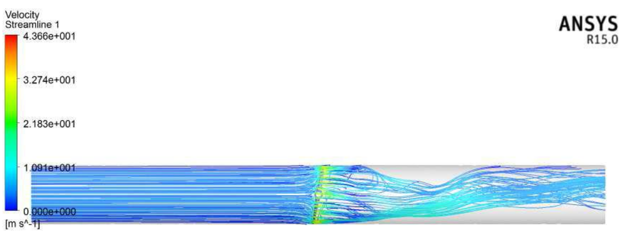
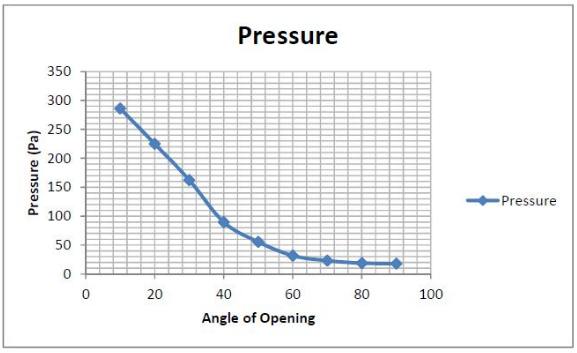
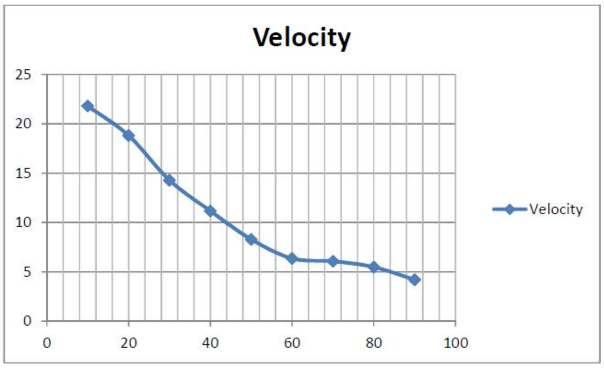


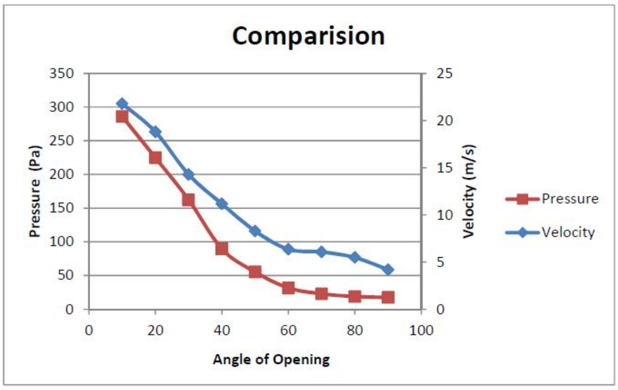
Figure 5.1 shows the velocity streamlines for 10 degrees valve disc opening.



Plot for velocity and valve angle of opening from CFD



Plot for Pressure and valve angle of opening from CFD



Comparison of Pressure and Velocity Contours from CFD

VI. CONCLUSIONS

It was concluded that for the pressure drop within and across the butterfly valve disc and only valve positions within 30-50 degrees were proficient of predicting the experimental value within error of 15%. For the hydro-dynamic torque, butterfly valve disc degree openings of around 20-80 degrees were able to give acceptable results. For the flow coefficient, valve degree positions of 20-90 degrees were able to give good results within for the loss coefficient, valve degree positions of 30-50 degrees gave exceptional results within error of 10%. For the torque coefficient, all valve degree positions were able to predict the setup values within error of 15%, with most within 10%. The refining guides for the performance aspects on the 10, 50, and 90 degree cases were all well in 5%, with exception of the hydrodynamic torque at the 90 degree open location which was at a value above 7%.

Versteeg and Malalasekera cite the main bases of error and ambiguity that are predominant in CFD. Causes of errors comprise: numerical errors. Numerical errors refer to round off errors, iterative junction. Coding errors refers to mistakes in the Ansys initialisation, which is inherent in unverified CFD code. User errors refer to human errors through incorrect analysis of boundary conditions. For this study, featured iterative convergence was achieved and double precision was used. The verified Ansys Fluent software was also performed by a CFD experienced individual. Furthermore, the grid convergence results using the same software showed that the error due to discretisation 75 was negligible compared to other probable sources of error. Thus, it is not supposed that numerical, coding and/or user errors are a significant source of error. Apart from human error uncertainty of ansys software and importing of solid model for analysis. Errors in solid models will reflect in meshing and refining analysis. Physical model uncertainties refer to discrepancies between real flows and CFD due to insufficient demonstration of physical or chemical processes (e.g. turbulence, convergence, etc.) or due to simplifying conventions in the displaying process (e.g. incompressible flow, steady flow). While it is difficult to release the amount of ambiguity due to any one factor, it is the author's confidence that the largest amount of ambiguity in the CFD simulations presented here arise from the turbulence modelling aspect, and correct representation of boundary conditions. In general, the performance factors were observed to have larger relative differences from the said values when the valve was at both lesser and advanced valve degree prologues. It may be that for very low valve positions, when the flow is restricted around the butterfly valve disc and causes to lower the flow velocity. Because most authors have replaced tabulated data from their studies on predicting butterfly valve performance factors, it is impossible to determine how well their effects equate to those of this study.
Only graphed outcomes showing an overall settlement were available, which can be misleading of the quality of the consequences and to their study. However, Song et al. and Chaiworapuek et al. brief mention seeing relative differences up to 50% for some flow performance factors. Song noticed this occurred at smaller valve angle openings.

A boundary condition of large alarm is the outlet boundary condition. the total length downstream of the buttery valve was around 14D. The simulation attempted to use around 12D with an assumed zero gradients out flow boundary condition. Knowing the exact upstream effects of the trial's control valve downstream of the valve before exiting to atmospheric conditions, is also difficult to enumerate. Ideally, if the shape of the downstream control valve and accompanying atmospheric discharge could be modelled in the replications, it would demonstrate ideal over the present technique used. However, due to limited information, this became a basis of ambiguity due to the assumptions implemented.

VII. ACKNOWLEDGMENT

We extend our sincere thanks to project guide and Assistant professor **Prof. Mrigendra Singh (Mechanical Department)**

REFERENCES

[1] Cohn, S.D., , \ Performance of Butterfly Valves," J. Instruments and Control Systems, 24.
[2] McPherson, M.B., \ Butterfly Valve Flow Characteristics," J. Hydraulics Division, 83(1).
[3] Sarpkaya, T., \ Torque Characteristics of Butterfly Valves," J. Applied Mechanics, 28(4).
[4] Addy, A.L., \An Investigation of Compressible Flow of Butterfly Valves," J. Fluids Engineering,107(4).
[5] Eom, K., \Performance of Butterfly Valves as a Flow Media," J. Fluids Engineering, 110(1).
[6] Kimura, T., Tanaka, \Hydrodynamic Characteristics of a Valve - Prediction of Pressure Loss Characteristics," ISA Trans., 34(4).
[7] Ogawa, K., and Kimura, T., \Hydrodynamic Characteristics of a Butterfly Valve - Prediction of Torque Characteristics," ISA Trans., 34(4).
[8] Huang, C., and Kim, R.H., 1996, \Three-dimensional Analysis of Partially Open Butterfly Valve Flows," J. Fluids Engineering, 118(3).
[9] Blevins, R.D., 1984, Applied Fluid Dynamics Handbook, Van Nostrand Reinhold, New York, NY.
[10] Lin, F., and Schohl, G.A., 2004, \ CFD Prediction and Validation of Butterfly Valve
[11] Hydrodynamic Forces," Proceedings of the World Water and Environmental Resources Congress.