

FLOW ANALYSIS OF BUTTERFLY VALVE USING CFD

Aniruddha V Kapre¹, Yogesh Dodia²

¹M.Tech IV Semester, Dept of Mechanical Engg, LNCT, Indore

²Asst Prof- Dept of Mechanical and Automobile Engg, LNCT Indore

ani_kapre@yahoo.com¹, yogesh.sgsits@gmail.com²

Abstract

A butterfly valve is a type of control valve which is used for isolating or diverting the flow. The working mechanism takes place from the disc. Function is similar to that of a ball valve, which allows for quick close and open systems. Butterfly valves are selected because of their lower cost to other valves manufacturing techniques, also lighter in weight, meaning less support is required. Rotating disc is placed in the center of the valve body. Disc can be operated by either manual handle or electrical actuators of various torques. The close-off pressure acting on the disc is directly proportional to the in-line pressure in pipe. The valve is selected according to the pressure and torque and pipe diameter, which allows lowering the damage the valve in operation.

Fluid flow is controlled through piping systems, which in turn are connected to the valves. The valve body comprises of disc, which also controls the flow of fluid through various angles of opening. The disc is connected in Centre through the stem which is connected to either handle or the rotating actuator. The design of the valve body and disc is of much important for linear or equal percentage flow. Nowadays the equal percentage flow is obtained by characterizing disc, which allows to maintain the equal percentage flow at different differential pressure across the pipe. Sometimes it becomes very critical to maintain the equal percentage flow; CFD can predict the flow performance and rate of discharge through the valve.

This paper focuses on the study of 8" and 20" valves performance factors and results obtained by analysis as well as from the load bearing capacity of the valve

However special thanks to Prof. Yogesh Dodia for his guidance to make this paper rational and justifiable.

Keywords: Butterfly valve, CFD, Stress Analysis, orthogonal array, optimization.

-----***-----

LITERATURE REVIEW

CFD is the most advanced form of analysis which gives the results of complete flow analysis. The flow may be compressible or incompressible and the boundary conditions may vary from single and multi-phase phenomena. This tool is used to insight the performance of our products without doing any experimental analysis. However the experimental analysis is also much important to validate the results.

INTRODUCTION

Butterfly valve is type of control valve which is used to regulate the flow of fluid through 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.

VALVE ATTRIBUTES

Many of the butterfly valves are identical in features such as the axis orientation, mounting methods and the disc. Therefore the installation of the valve w.r.t disc becomes easier and this is commonly termed as seating direction of the valve. The installation methods affect the flow characteristics and the performance. For optimized flow these valves can be installed in any of the directions upstream or downstream. When the disc is tightly closed with seat the upstream denotes the valve is fully closed. On the other hand if disc is at 90 degrees to the valve or parallel to the flow the downstream denotes the valve is open. Key diagram for the butterfly valve is shown in fig.1. The EPDM seat ensures the tight shut-off of valve when the valve is closed and hence the valve becomes leak proof which to be followed as per Air tight leak proof EN12266-1. Advanced profile of the disc can be incorporated for narrowing the inline pressure effect on the valve. The primary

seal is achieved by an interference-fit of the molded seat flat with the disc hub. Many stringent and extensive HVAC flow media with wide and circular arc flange sealing surface, which can be suitable for all flange connection standard as ISO, JIS without additional gasket to fit easily and efficiently.

Many of the butterfly valves are identical in features such as the axis orientation, mounting methods and the disc. Therefore the installation of the valve w.r.t disc becomes easier and this is commonly termed as seating direction of the valve. The installation methods affect the flow characteristics and the performance. For optimized flow these valves can be installed in any of the directions upstream or downstream.

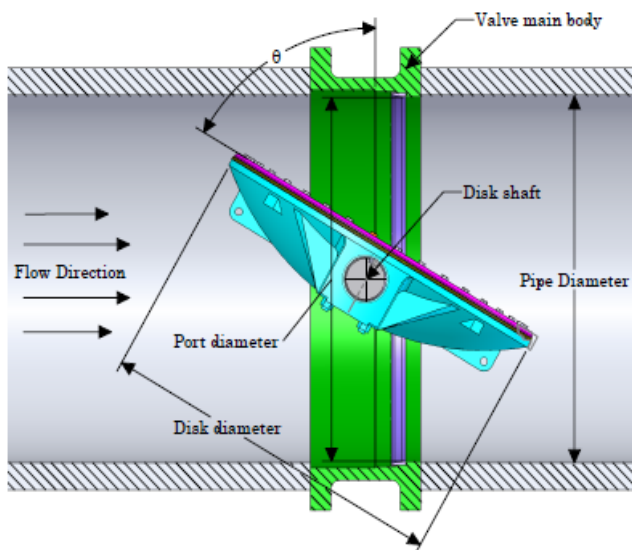


Fig1:- Cross-section of a 48-inch butterfly valve installed in a pipeline in the seated downstream position and open at an angle θ

When the disc is tightly closed with seat the upstream denotes valve is fully closed. On the other hand if disc is at 90 degrees to the valve or parallel to the flow the downstream denotes the valve is open. Key diagram for the butterfly valve is shown in fig.1. The EPDM seat ensures the tight shut-off of valve when the valve is closed and hence the valve becomes leak proof which to be followed as per Air tight leak proof EN12266-1. Advanced profile of the disc can be incorporated for narrowing the inline pressure effect on the valve. The primary seal is achieved by an interference-fit of the molded seat flat with the disc hub. Many stringent and extensive HVAC flow media with wide and circular arc flange sealing surface, which can be suitable for all flange connection standard as ISO, JIS without additional gasket to fit easily and efficiently.

CFD SETUP

General purpose CFD software SIM SCALE is used to investigate the model. However the 3-D model is created in Solid Edge V19.0. The file is exported to cfd software for analysis. Here the model is converted to STEP file. The accurate cylindrical parts are added to the part for proper simulation. The simulation is done for upstream and downstream flow through the valve. Meshing of the part and

added cylindrical parts has to be done properly. The boundary conditions such as flow rate, fluid properties, valve angle opening etc depends on the mesh geometry which also depends on the mesh quality and the iterations of mesh geometry. Moving to this we can confirm the performance and grid convergence, iterations required for the perfect mesh geometry. The predicted performance factors for flow and mechanical properties are then studied for efficient flow analysis. In this case the simulations are carried out for angle from 10 degrees to 90 degrees in ten degrees intervals. Extra two simulations are carried out for 10 and 50 degrees with coarser meshes to analyze the mesh quality. The meshing is done with Snappy Hex Mesh for internal flow.

Governing equation for fluid dynamics also referred as incompressible isothermal fluids in CFD includes the Navier-Stokes equations given in conservative form. These equations are a simplification of momentum equations and behavior of the fluids is line with Newtonian fluid.

For incompressible flow:

$$\text{Continuity: } \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$\text{Navier Stokes equation: } \rho \frac{D\vec{V}}{Dt} = \rho \vec{g} - \nabla P + \mu \nabla^2 \vec{V}$$

Where

$$\frac{D}{Dt}(\vec{V}) = \frac{\partial}{\partial t}(\vec{V}) + u \frac{\partial}{\partial x}(\vec{V}) + v \frac{\partial}{\partial y}(\vec{V}) \quad (2)$$

$$\nabla^2(\vec{V}) = \frac{\partial^2}{\partial^2 x}(\vec{V}) + \frac{\partial^2}{\partial^2 y}(\vec{V}) \quad (3)$$

Also certain assumptions were made for conduct the analysis

- Common assumptions are:
 - Incompressible $\rightarrow \rho$ and μ are constant
 - Steady state $\rightarrow \frac{\partial \vec{V}}{\partial t} = 0$ (time independent)
 - Fully developed $\rightarrow \frac{\partial \vec{V}}{\partial x} = 0$ (space independent)
- Other assumptions may include:
 - Neglect gravity $\rightarrow \vec{g} = 0$

$$\text{No pressure gradient } \rightarrow \nabla P = 0$$

The equations in the boundary layer theory are simplified form of Navier Stokes equations and provides the calculated means of Cform. Earlier the boundary layer theory predicts the asymptotic form of Navier-Stokes equations for high Reynolds number for slender parts. The NV equation for second order partial differential equation gives the simplified solutions. But here we restrict our study for CFD analysis using these equations for our analysis.

MESH

These continuity equations are required to convert to volumetric cells of mesh which comprises of minute vertex and flat faces. This is the new method of solving the equations by numerical techniques. This analysis represents the mathematical description of the mesh and the part geometry of the intended problem to be solved.

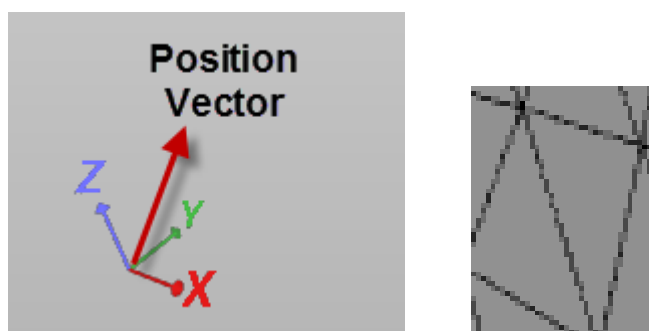


Fig 2. Illustration of the components of a designed volume mesh.

GEOMETRY IMPORT

The geometry is modelled in solid edge for 3-D analysis. The model is assigned as part and later converted to STEP file for further analysis in Simscale. The butterfly valve is checked for various angles for flow analysis. A finer level of mesh geometry will preserve the part geometry and the CAD file for further analysis. For this analysis the pipe length is assumed to be 5 times of the valve diameter. Our main concern is to study the flow analysis when it reaches the valve disc and then flow pattern changes.

Firstly the mesh is to be created for the valve assembly and mesh geometry is to be imported for CFD analysis. The Mesh is created using Snappy Hex Mesh for internal flow with Single order 16 cores processor and Moderate finish Mesh.

BOUNDARY CONDITIONS

The boundary conditions define the working medium and the main constraints in the flow path. Here we have assumed these conditions as inlet velocity of fluid, outlet velocity of fluid and wall.

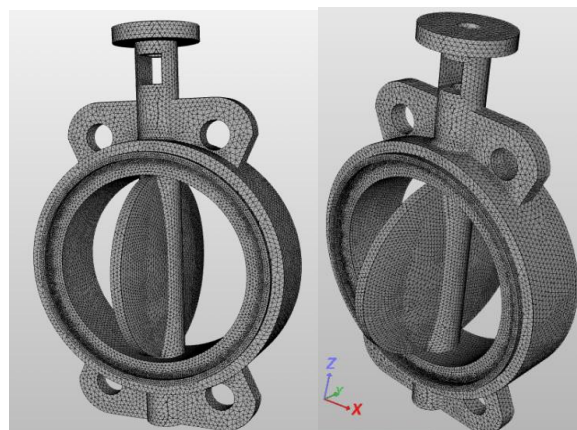


Fig 3. Mesh Geometry of the valve DN500 & DN200

For more study we can discuss the following momentum equation for finer analysis.

Momentum Equation

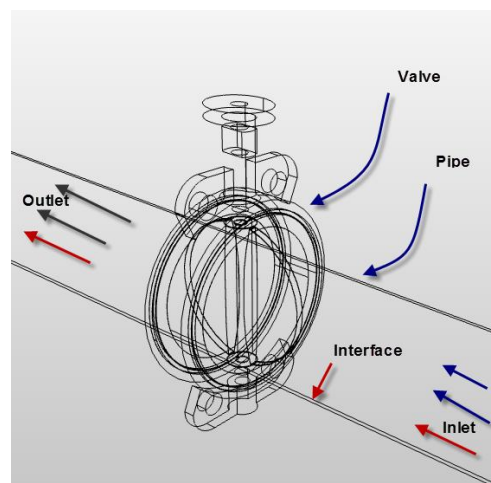
- x - momentum:

$$\rho \frac{D}{Dt}(u) = \rho g_x - \frac{\partial P}{\partial x} + \mu \nabla^2(u) \quad (4)$$

$$y - \text{Momentum: } \rho \frac{D}{Dt}(v) = \rho g_y - \frac{\partial P}{\partial y} + \mu \nabla^2(v)$$

Further the displacement of valve disc is in the direction hence $d_z = \theta = 0$ to 90 degrees. And load on the valve seat and valve actuator interface is in the Y direction, and hence $f_y = 5000N$.

The model for the flow equations which includes the velocity, turbulence equations and pressure equations; also can be changed from boundary conditions for finer mesh quality. The concept of these equations is to maintain the parametric relation between momentum and continuity equations which is achieved by predictor-corrector approach. Options and working features of boundary conditions are second order for convection scheme, pressure of min 1000Pa. boundary diffusion enabled, and delta-V dissipation disabled.



Topological entity for the valve assembly is considered on the face and not the volume. Hence the load is applied on the faces and following fig4 shows the load conditions.

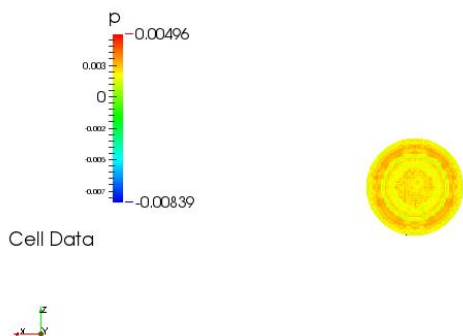


Fig 4. "Plot imported from Simscale after analysis"

RESULTS

CFD simulations done were carried out using the criteria mentioned in earlier analysis. The flow vectors generated by the simulations were also analyzed, The performance factors were also calculated and whole flow analysis is carried out for 3 different angles of disc 10, 50 and 90 degrees.

Following three figures indicates the flow analogy through pipe at three different angles

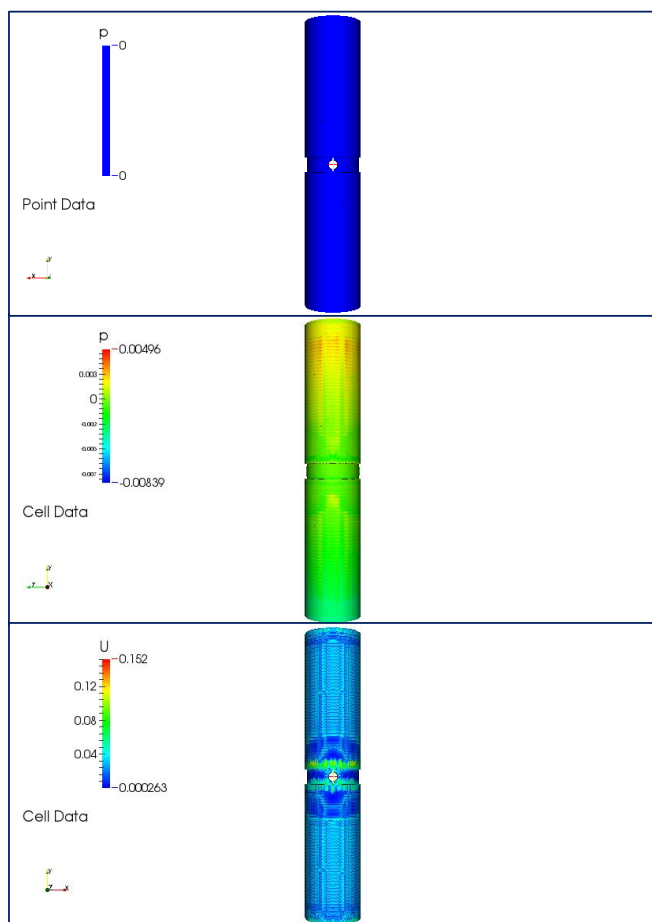


Fig 5. Flow pattern through three different angles (90°, 50°, 10°)

The flow at different angles of the disc rises the different flow pattern in the form of vortices which are formed along the pipe wall. The absolute pressure and velocity legend represents the disc area and the streamlines generated as per the Reynolds number. The streamline image shown in Fig7 depicts the flow parallel to the pipe wall surface which indicates the streamlines along valve opening disc and pipe wall. After fluid flow through valve disc and boundary layer separation; this causes the very high turbulent and swirling behavior. Henderson et al. [3] also noted this behavior and the presence of a strong pair of vortices behind butterfly valves as seen in Fig. 6 & 7. Also we can predict the flow pattern in X, Y & Z direction as shown in Fig 6.

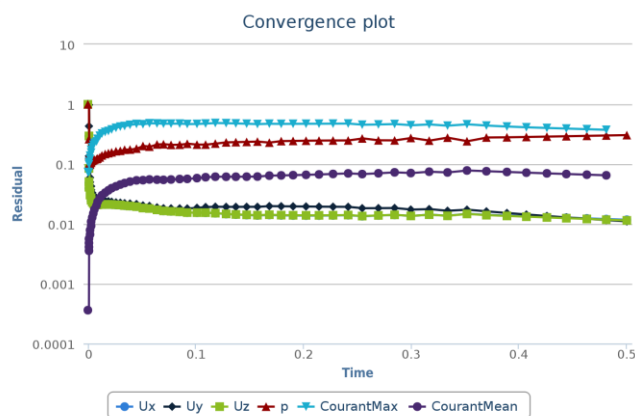


Fig 6. Convergence plot showing the flow pattern

During the flow the valve undergoes several loading conditions and dynamic loading . Mechanical loading, thermal stress, bolt tension, pressure conditions and rotational acceleration are just some of the factors that will dictate strength requirements for materials and designs.

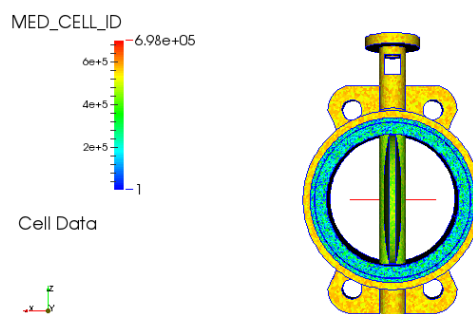


Fig 7. Thermal Stresses on Valve seat at 90° of disc.

CONCLUSION

This CFD analysis has validated the numerical analysis along with flow pattern through the butterfly valve. The results derived are streamline with transient state flow analogy which is in downstream from the valve. But the flow beyond the specified pressure range and for various angles becomes unsteady. The flow pattern with set boundary conditions and parameters are turbulent but the numerical analysis may vary from different boundary conditions. This analysis highlights

the need for better measurements that may include mass flow rate. By this the CFD analysis and the simulations can be achieved at far more better mesh quality. Only thing we can change to achieve finer results is to use fewer processors. Now we can validate the results achieved through this analysis.

REFERENCES

- [1] CFD prediction and validation Butterfly valve Hydrodynamics forces, Fangbiao Lin , 2004
- [2] Computational Fluid Dynamics Analysis of Buttery Valve Performance Factors, Adam Del Toro, Utah State University, Logan USA.
- [3] Henderson, A.D., Sargison, J.E., Walker, G.J., and Haynes, J.H., 2008, \A Numerical Prediction of the Hydrodynamic Torque Acting on a Safety Buttery Valve in a Hydro Electric Power Scheme," WSEAS Trans. on Fluid Mechanics, 1(3), pp. 218{223.}
- [4] Flow characteristics and performance evaluation of butterfly valves using numerical analysis S Y Jeon, J Y Yoon and M S Shin, 2010