Flow Analysis of Butterfly Valve Using CFD

Arun Azad[1], Deepak Baranwal[2], Rajeev Arya[3], Nilesh Diwakar[4]

[1] *PG Research scholar Department of Mechanical Engineering TIEIT, Bhopal*
[2] *Research scholar Department of Civil Engineering MANIT, Bhopal*
[3] *Professor & Dean PG Department of Mechanical Engineering TIEIT, Bhopal*
[4] *Professor & HOD Department of Mechanical Engineering TIEIT Bhopal*

Abstract: *Butterfly valves are widely used in hydro power plants to regulate and control the flow through hydraulic turbines. That's why it is important to design the valve in such a way that it can give best performance so that optimum efficiency can be achieved in hydraulic power plants. Conventionally that the models of large size valves are straight in the laboratory to determine their performance characteristics. This is a time consuming and costly process. High computing facility along with the use of numerical techniques can give the solution to any fluid flow problem in a lesser time. In this research work flow analysis through butterfly valve with aspect ratio 1/3 has been performed using computational software. For modelling the valve ICEM CFD 12 has been used. Valve characteristics such as flow coefficient and head loss coefficient has been determined using CFX 12 for different valve opening angle as 30°,60°,75°, and 90° (taking 90°as full opening of the valve) for incompressible fluid. Value of head loss coefficient obtained from numerical analysis has been compared with the experimental results.*
Key Words: *Butterfly valve, CFD flow analysis, simulation, valve opening.*

I. Introduction

Butterfly valve is very versatile component for use both as shut off and throttling valve in water system. Butterfly valve has simple mechanical assembly, and a small flow resistance in a fully open position, butterfly valves provide a relatively high flow capacity. They are the best suited for relatively low pressure flow. Generally, the fluid friction coefficient is low and also the build up is usually minimum because of the Butterfly valve is operated with a quarter turn (like the ball valve). Manual Operation may be through lever or gear. The position of the disc is noted by an indicator on the gearbox or from the position of the handle. The first attempt at collecting and collating the published data concerning butterfly valves was probably made by Cohn in 1951. Experimental studies on butterfly valve flow characteristics have been conducted by Addy et al in 1985.The results of numerical simulation of flow characteristics including both velocity and pressure calculations are presented in literature. Study on hydrodynamic torque of the butterfly valve has been conducted by Morris and Dutton in 1989.

II. Technical Requirements Of Butterfly Valve

Main purpose of the valves installed ahead of turbine is to close the penstock while the turbine is not in operating condition. Valve must possess high degree of reliability and durable strength at closed position so that trouble free operation can take place. It should also ensure the possibility of carrying out repair works, revisions of turbines, pumps, pressure penstock and at the same time to create lowest resistance to flow. Valve must close at velocity of water occurring during emergency cases (rupture of penstock or runway speed of turbine).Maximum time for closure of the valve when it is installed before the turbine pump is 1-3 minutes and depends upon the permissible time of operation of generator at run way speed and motor of the pump in generator regime. If the valve is installed at the beginning of the penstock, then its closure time will be in the range of 30-120 seconds. Time of closing of valve on sluice and irrigation schemes is determined from the operational conditions of these constructions. Minimum time of emergency closure is determined as per permissible value of hydraulic impact on penstock. Operation of spherical and b.f. valve is depend upon the extreme positions of the rotating part while in case of cylindrical valve it is accomplished at any position including extreme positions for regulating discharge of liquid through them is zero to maximum possible discharge. Servomotors of various designs are used to rotate the rotating element, by mechanical or electro mechanical drive.

III. Objective

Butterfly valves are required to have high performance characteristics and better precision as they are used as shut off valves. The characteristics of a valve i.e. head loss characteristic, torque characteristics and force characteristics of butterfly valve is determined conventionally through tests. If the test valve is of large size, scale model of valve is tested to determine its characteristics. Analysis of flow characteristic experimentally is a hectic and not very precise work. Exact theoretical analysis of flow through complex geometry is very difficult with the use of high speed computers, and the numerical techniques, the flow analysis can be made using CFD.That's why flow analysis is to be performed using simulation software. Objective of this research work is to determine head loss coefficient and flow coefficient, for different valve opening and discharge value. At present almost every industry is using software for analysis, Butterfly valves are used in various power plants, so it has wide scope, this research work will be beneficial for all those industries that are using butterfly valve. Computational fluid dynamics is a tool to carry out numerical flow simulation for predicting the flow behaviour within a specific domain by numerical solution of governing equations to acceptable accuracy. Computational fluid dynamics is becoming very useful approach for engineering design and analysis because of improved numerical method and at the same time, it saves time and energy of experimental work. In this thesis, flow simulation has been carried out for double disc butterfly valve using Ansys ICEM CFD and CFX. The geometric modelling is done for four angular position of valve disk to assess the head loss and discharge coefficients Characteristics of valve.

IV. Computational Fluid Dynamics

CFD is a computational technology that enables to study the dynamics of matters that flows. CFD is predicting what will happen, quantitatively, when fluids flow even with the complications of simultaneous flow of heat, mass transfer, phase change, chemical reaction, mechanical movement, stresses in and displacement of immersed or surrounding solids. CFD include expressions for the conservation of mass, momentum, pressure, species and turbulence. Navier-Stokes equation given by Claude Louis Marie Henry Navier and the George Gabriel Stokes. Which defines any single-phase fluid flow, is the fundamental bases of all CFD problems.CFD software is based on sets of very complex non-linear mathematical expressions that define the fundamental equations of fluid flow, heat andmaterials transport? These equations are solved iteratively using complex computer algorithms embedded within CFD software. Outputs from CFD software can be viewed graphically in colour plots of velocity vectors, contours of pressure, lines of constant flow field properties, or as "hard" numerical data and X-Y plots. In CFD it is possible to directly solve the Navier-Stokes equations for laminar flows and for turbulent flows when all the relevant length scales can be resolved by the grid (a direct numerical simulation). In general however, the range of length scales appropriate to the problem is larger than even today's massively parallel computers can model. In these cases, turbulent flow simulations require the introduction of a turbulence model. Large eddy simulations (LES) and the Reynolds-averaged Navier-Stokes equations (RANS) formulation, with the k-ε model or the Reynolds stress model, are two techniques for dealing with these scales.

V. Boundary Conditions

Here inlet, outlet and other boundary conditions are defined. In this analysis in pipe domain inlet and outlet boundaries has been created and subjected to various Conditions.

1. Inlet boundary condition
In this location is selected as inlet of the pipe, flow is subsonic, mass flow rate has been set different for different opening angles. Flow direction is normal to the boundary and turbulence is set at medium (intensity =5%)

2. Outlet boundary condition
In this location is selected as outlet of the pipe, flow is subsonic, average static pressure has been taken as 1 atm.
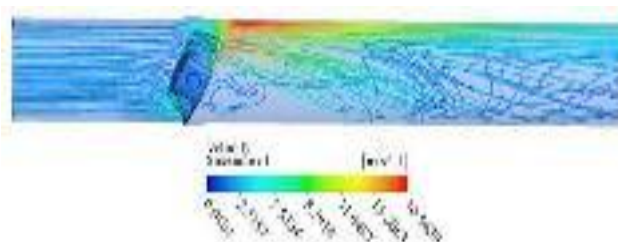
VI. Velocity Distribution

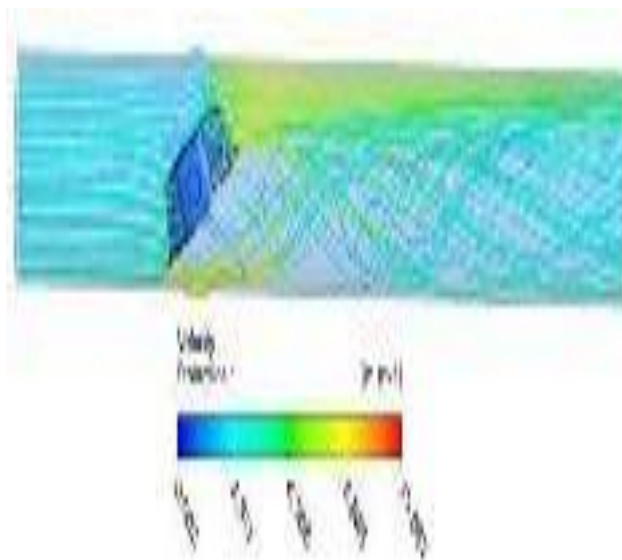Fig.1 streamlines of flow through butterfly valve at 30°opening.

Fig.2 streamlines of flow through butterfly valve at 60°opening.
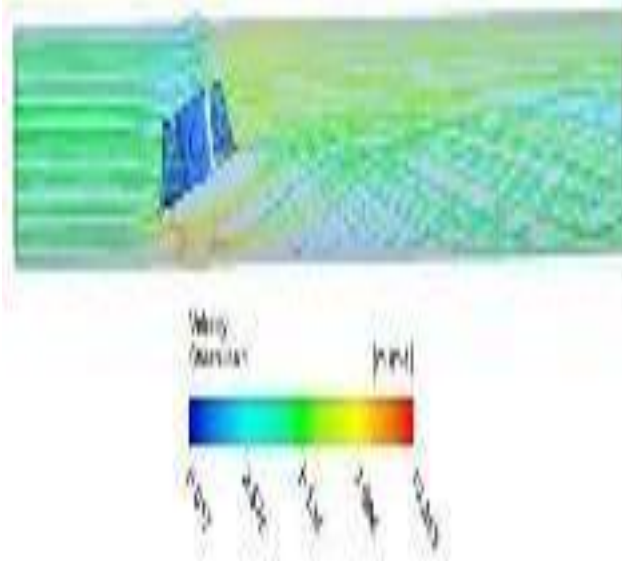

Fig.3 streamline of flow through butterfly valve at 75°opening.
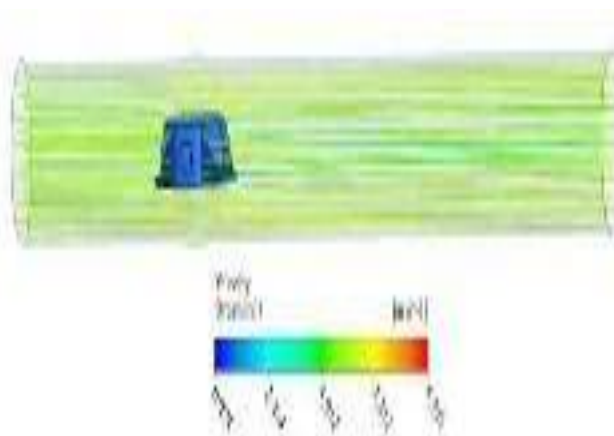

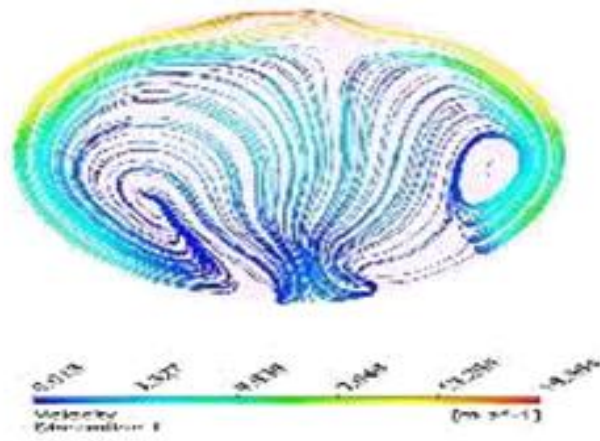Fig.4 streamline of flow through butterfly valve at 90°opening

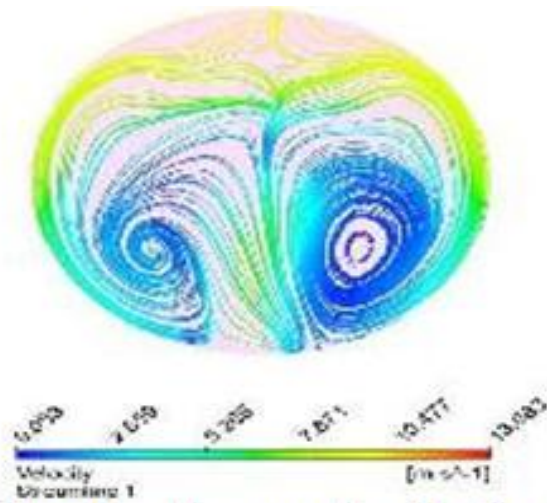Fig.5 streamlines at the cross section of the pipe at downstream for 30°.

Fig.6 streamlines at the cross section of the pipe at downstream for 60°.
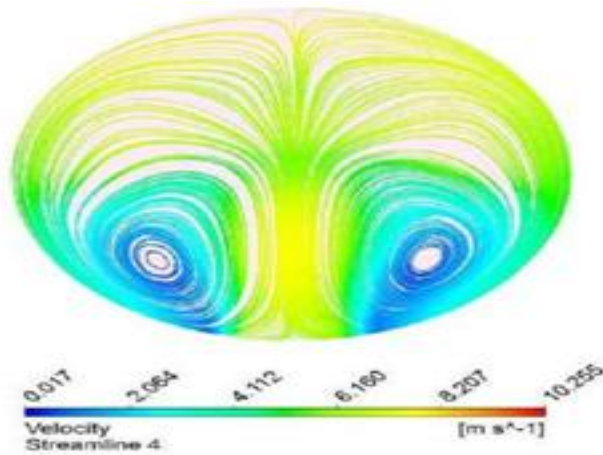

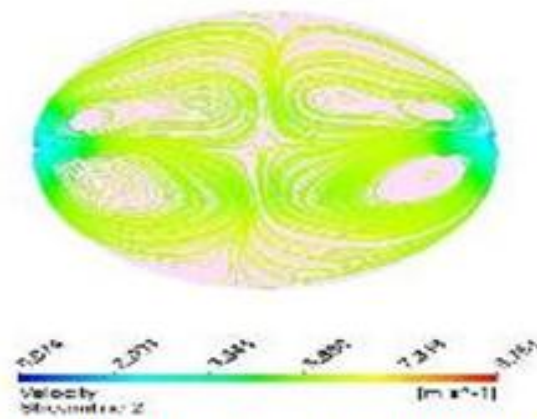Fig.7 streamlines at the cross section of the pipe at downstream for 75°.

Fig.8 Streamlines at the cross section of the pipe at downstream for 90°.
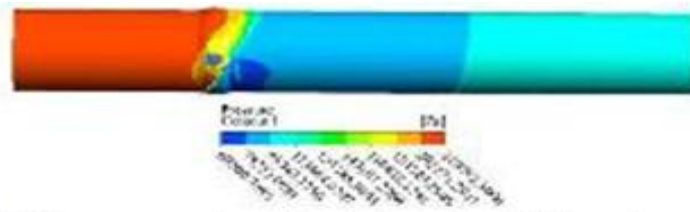
VII. Pressure Distribution
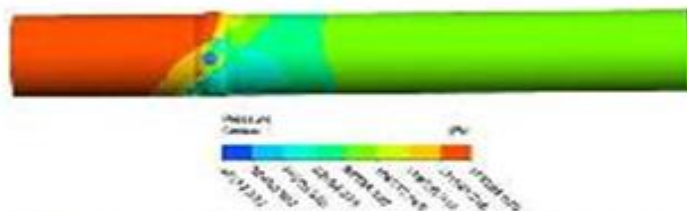

Fig.9 pressure contour of butterfly valve at 30° opening
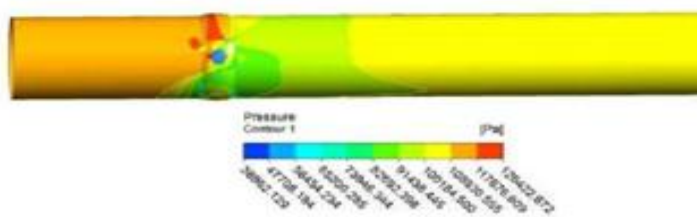

Fig.10 pressure contour of butterfly valve at 60°opening

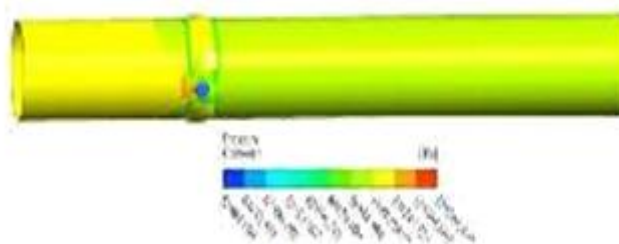Fig.11 pressure contour of butterfly valve at 75°opening

Fig.12 pressure contour of butterfly valve at 90°opening
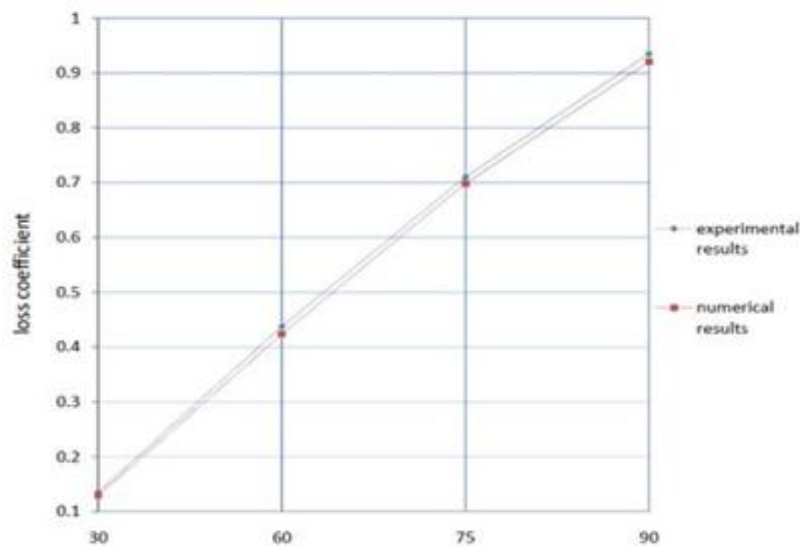
VIII. Result

Fig.13 Comparison of numerical and experimental results

IX. Conclusion

During this research work, analysis of flow through Butterfly valve has been done to determine the performance characteristics by CFD analysis and based on the simulation results, following conclusions are drawn:

- Velocity at upstream as well as downstream is increasing with the increase in opening angle.
- Streamlines at the downstream side of the door is becoming uniform with the increase in opening angle. This indicates that the disturbances are reduced for higher valve opening angle.
- Total pressure at upstream is decreasing while at downstream side of the door it is increasing with the opening angle. This indicates that the pressure imbalance is reducing on the two sides of the door with increase in angle of opening.
- Streamlines at the cross section of the pipe shows that vortices are shifting towards downward direction and becoming clearly visible with increase in opening angle.
- At small angle of opening, there are vortices formed behind the door. The turbulence in the flow is also high. These are causing large form drag and a high amount of total loss. Relative loss coefficient is decreasing, that's why head loss coefficient is increasing with the increase in valve opening angle. It shows that head loss coefficient is a strong function of opening angle .
- Flow coefficient is increasing with the valve opening and having maximum value 0.1630 for fully open condition .
- Numerical results are matching with the experimental results very closely, thus conforming the present CFD analysis .

REFERENCES

[1]. A. D. Henderson, J. E. Sargison, G. J. Walker and J. Haynes, "A numerical study of the flow through a safety butterfly valve in a hydro electric power scheme",16th Australasian fluid mechanics conference, December 2007.
[2]. Fangbiao Lin and Gerold A. schohl,:-"A CFD prediction &validation of butterfly valve hydrodynamic forces", ASCE Conf. Proc. doi:10.1061/40737(2004)232 Critical Transitions in Water and Environmental Resources Management, World Water and Environmental Resources Congress, 2004
[3]. Gerhart Penninger, Helmut Benigni" New design of butterfly valves for modern pump-storage power plants" Voith Siemens Hydro Power GenerationGmb H&Co. KG and Institute for Hydraulic Fluid machinery, Engineering &Design – TUK Technical University of Graz.
[4]. Hui-xiong Wan.Jun Fang and Hui Huang "Numerical simulation on a throttle governing system with hydraulic butterfly valves in a marine environment"Harbin engineering university & spriger-verlay Berlin hedelberg, 2010.
[5]. Janusz WOJTKOWIAK, Czeslaw oleskowkz-popiel investigation of butterfly valve flow characteristics"
[6]. Jeon S Y, Yoon J Y, Shin M S."Flow characteristics and performance evaluation of butterfly valves using numerical analysis", 25th IAHR Symposium on Hydraulic Machinery and system.
[7]. M. S. Kalsi, B. Eldiwany, Vinod Sharma, Aaron Richie "Effect of butterfly valve disc shape variations on Torque Requirements for power plant application", Eight NRC/ASME Symposium on valve and pump testing, Washington, D.C, July 2004

[8].   Kim S. W., Kim J. H., Choi Y. D., Lee Y. H. ,"Flow characteristics of Butterfly valve and CFD" Fifth international conference on fluid mechanics, Aug 2007,shanghai,Chaina.

[9].   Lin Wang, Xueguan Song, Youngchul Park, "The improvement of large butterfly valve by using numerical analysis method" 6th WSEAS international conference on fluid mechanics, 2009.

[10]. Park Youngchul and Song Xueguan "Numerical Analysis of Large Diameter Butterfly valve" springer science business Media, 2009.

[11]. V.Caille, J. Laumonier,"Effect of periodic aerodynamic pulsation on flow over aconfined butterfly valve", journal from spriger-verlog, 1998

[12]. Weerachai Chaicorapuek: -"The engineering investigation of the water flow past the butterfly valve" June 2007,

[13]. Xue Gguan Song, Young Chul Park, "Numerical Analysis of butterfly valve prediction of flow coefficient and hydrodynamic torque coefficient", world congress on engineering and computer science, October 2007, San Francisco, USA.

[14]. Zachary Leutwyler, Dalton Charls"A computational study of torque and force due to compressible flow on a butterfly valve disc in mid stroke position" September-2006,J fluid engg, volume 128,issue 5,1074

[15]. Zachary Leutwyler, Charls Dalton "A CFD study of flow field, resultant forced aerodynamic torque on a symmetric disc butterfly valve in a compressible Fluid" may-2008, j.pressure vessel techno l volume 130, issue 2012-13 02

[16]. A Thesis submitted by Mukesh Kumar Meena,"Constructional and operational aspects of main inlet valve of hydro power plants", civil engineering Department, MANIT Bhopal, February 2005.

[17]. Ashok Kumar, Jsv Sharma, "Site Experience of spherical and Butterfly valve "National seminar on Gates and Valves, July 1984.

[18]. http://forum.bulk-online.com/showthread.php?7088-How-amp-Why-Are-Butterfly-Valves-Used

[19]. http://www.1butterflyvalve.com/technology-1.html (history)

[20]. http://scholar.google.co.in/

[21]. www.cfd-online.com
       CFX-12.0, User Manual from http: //www.ansys.com.

[22]. Jun-Oh Kim, Seol-Min Yang, Seok-Heum Baek, Sangmo Kang" Structural Design Strategy of Double-Eccentric Butterfly Valve using Topology Optimization Techniques" World Academy of Science, Engineering and Technology 66 2012.

[23]. Abhijeet T. Bhosale , A.S.Dhekane "Finite Element Analysis of based Optimization Butterfly valve disc" International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 10, October - 2013