Proceedings of:
Current Research in Hydropower Technologies, CRHT X
Kathmandu University, 2020
CRHTX-28

Design and Optimization of Butterfly Valve Disc Using Numerical Simulation

Chhantyal Bikki[1*], Kafle Ranjeet [2], Tamrakar Smarika [2]

[1]Department of Mechanical Engineering IOE Pulchowk, Nepal*

[2]Mechanical Engineer, NEA Engineering Company Ltd, Trade Tower, Thapathali, Kathmandu, Nepal

** Corresponding author (chhantyalbikki@gmail.com)*

ABSTRACT

This paper describes the numerical method to optimize the butterfly valve used in hydropower projects for flow and pressure regulation, safety, maintenance and shut-off purposes. The main objective is to optimize butterfly valve disc based on stress criteria, head loss and weight, where the parameters taken to optimize the model are disc thickness, stiffener height and stiffener thickness. Main inlet valve under study is of size DN 2800, placed at a gross head of 169m and flow rate of 63.5 m^3/s. Initially, sizing of the valve disc is based on preliminary calculations and experience. Three levels of designs for each parameter are set. This resulted in twenty-seven cases for analysis. However, Taguchi Orthogonal Array is applied, which reduced to nine different cases for optimization. ANSYS Space claim is used to prepare the 3D model and determine the weight of the geometry. Head loss and flow distribution in fully open condition between the upstream and downstream, each of which is ten times the nominal valve diameter is calculated using ANSYS Fluent. Then, the structural and deformation of the valve disc at fully closed position is achieved using the ANSYS Structural. Finally, the optimized model is selected based on structural stress, deformation and minimum head loss.

Keywords: Butterfly valve, ANSYS Fluent, ANSYS Structural, Taguchi Orthogonal Array, Disc Thickness, Stiffener

INTRODUCTION

Water is the source of energy for hydroelectric power plants where the kinetic and potential energy is converted to mechanical energy through turbine and generator converts it to electrical energy. In the hydropower project, valves are typically used as control devices in order to regulate the flow of water. It stops the water entry in case of emergency. Commonly, main inlet valve is used in multi-unit plants where one penstock feeds more than one unit for unit isolation. Valve assists in inspection of penstock without dewatering the entire head race tunnel, acting as an isolating device. Generally, butterfly valves are used in low head applications up to 200 m.

Butterfly valve of size DN 2800, placed at a gross head of 169m, used to regulate flow rate of 63.5 m^3/s is under study. The objective of this study is to optimize the valve disc based on stress criteria, head loss and weight.

METHODOLOGY

Valve disc of material alloy cast steel is selected. The mechanical properties of the material is shown in table 1.

Table 1 Valve Disc Mechanical Properties

S.N.	Properties	Values
1	Yield Strength [MPa]	415
2	Poisson's ratio	0.303
3	Density [kg/m^3]	7800

At closed condition, the valve disc should bear hydrostatic pressure as well as surge pressure. The valve is designed to bear a total pressure equal to 253.5 MPa which is 1.5 times of gross head. In order to withstand this pressure, a normal circular plate disc of thickness is calculated to be 173 mm. This, however, accounts to 8.192 tons weight. The valve is optimized for structurally sound and minimum head loss.

Valve is optimized with three parameters i.e. valve disc thickness, rib thickness and rib height. Initially, valve sizing is done based on manufacturer's data and general consideration. Each parameter is associated with three levels of design which is shown below in table 2.

Table 2 Valve Parameters and Associated Values

S.N.	Parameter	Values
1.	Valve disc thickness	173 mm
		183 mm
		193 mm
2.	Rib thickness	50 mm
		60 mm

		70 mm
3.	Rib Height	670 mm
		760 mm
		850 mm

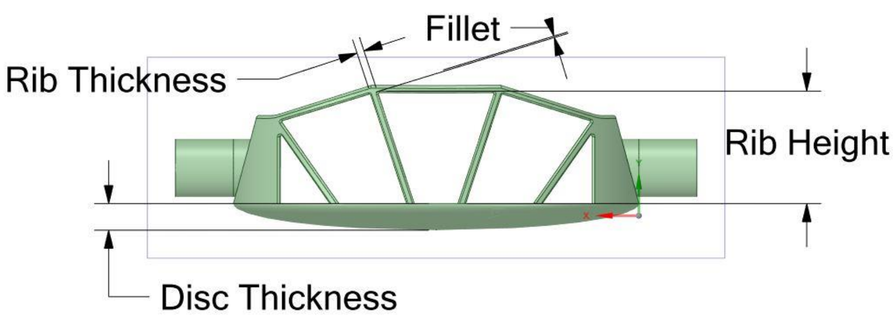

Figure 1 Valve Disc Parameters

Three design values set for two different parameters resulted in twenty-seven different cases for analysis. Taguchi Orthogonal Array [1] is applied as shown below to reduce the number of cases to nine models.

Table 3 Nine Models for Simulation

Model No.	Disc Thickness	Rib Thickness	Rib Height
1.	173 mm	50 mm	670 mm
2	173 mm	50 mm	850 mm
3	173 mm	60 mm	760 mm
4	183 mm	50 mm	760 mm

5	183 mm	60 mm	850 mm
6	183 mm	70 mm	670 mm
7	193 mm	50 mm	850 mm
8	193 mm	60 mm	670 mm
9	193 mm	70 mm	760 mm

STRUCTURAL SIMULATION

Structural simulation was performed in selected nine models in Table 3. The commercial software ANSYS Structural was used for the structural simulation. The maximum von misses stress and maximum deformations were observed for nine models. The model for the structural simulation was prepared in space claim 19.2. The rib thickness, rib height and disc thickness were varied for different model. The disc is elliptic in nature so, half of the ellipse in mid plane is revolved 360 degree. The fillet of 10 mm is done to avoid the stress concentration. The valve is divided into four parts. The shaft is 400 mm equal to the diameter of the shaft.

The valve is meshed with tetrahedral mesh with different sizing. The mesh independence in check in first model using total deformation as convergence parameter. It was found out that body sizing of 40 mm gives the accurate result within 1% convergence criteria.

Analysis and Setting

The valve experiences the hydrostatic pressure equivalent to head of 169 m. This head further increases in case of surging so, for static structural simulation head equivalent to 253.5 m is considered.

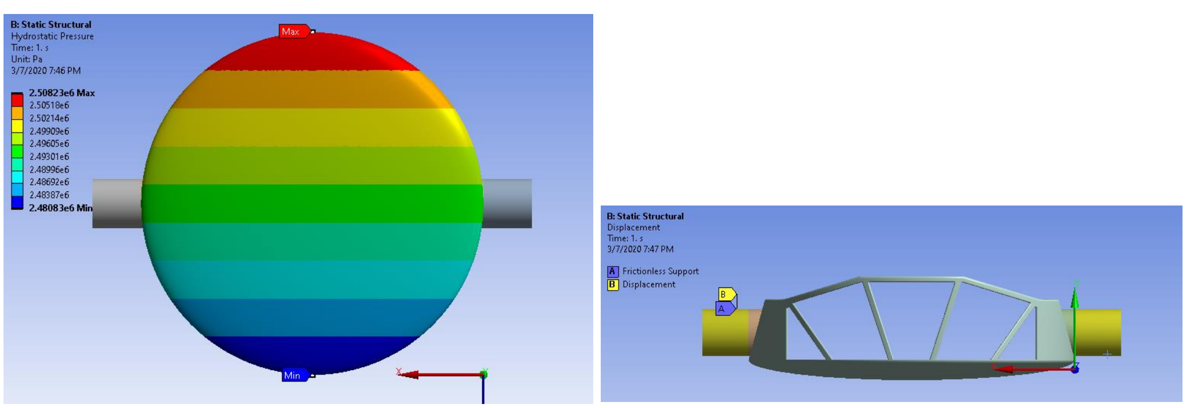

Figure 2 Boundary Condition

In the shaft cylindrical surface, the frictionless support is put to restrict motion in radial direction. Also, the displacement in x and y is set to constant 0 and displacement in z direction is set to free [2].

Results

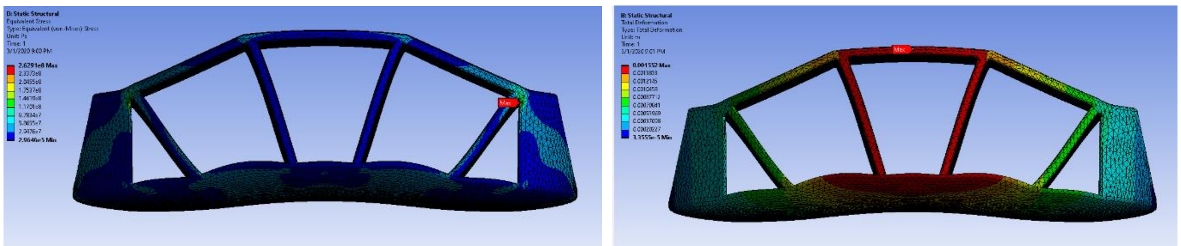

Figure 3 Stress (left) and Total Deformation (right)

The maximum value of the stress is 262.91 MPa as in the transition shown in the figure 3. Also, the maximum deformation is 1.552 mm at the mid-section of the upper rip. The deformation is also almost the maximum value in the middle of the disc. The maximum stress is less than yield stress, so it is safe to operate at the head. And the maximum deformation is less than 3 mm. Since, the seal is pre compressed by 3 mm, the leakage of water is prevented.

Table 4 Comparison of Stresses and Deformations

Model No.	Maximum Stress	Factor of Safety	Total Deformation
1	262.91	1.578	1.552
2	253.55	1.636	1.401
3	245.77	1.688	1.112
4	274.81	1.510	1.356
5	256.43	1.618	1.290
6	229.34	1.809	1.258
7	281.33	1.475	1.436
8	206.72	2.007	1.313
9	162.82	2.548	1.054

For all the models, the maximum stress is less than yield stress and total deformation is less than 3 mm, hence the model will be selected on the basis of minimization of head loss.

CFD SIMULATION

CFD simulation of Butterfly valve of different model was performed to calculate the head losses and flow distribution. The commercial software ANSYS Fluent was used for the CFD simulation.

The model prepared for the structural simulation in ANSYS Space claim 19.2 was subtracted from the pipe of constant diameter 2800 mm to create fluid domain. The pipe was extended two times the diameter in the upstream and 8 times the diameter of the pipe in the downstream [3]. This represents the simulation of butterfly valve with 90 degree opening without valve body and shaft for simplicity.

Figure 4 Fluid domain for CFD of Main Inlet Valve

For the mass flow rate of 63500 kg/s in the outlet with 2800 mm diameter, the Reynolds's number for the flow in all cases is approximately 3.15e7. It is in the turbulent flow region. So, the Shear Stress Transport (SST) k-w turbulence model was used for the simulation [4]. This model solves the complex fluid flow with k epsilon model in free shear region and with k-w model in near wall region. It considers the transport of shear stress to predict accurate fluid flow for high pressure gradient flows.

The first layer thickness of mesh near wall should capture the high gradient of velocity in the near wall region. This is characterized by the y+ value. The y+ value for the SST k-w model should be in between 200 to 500. Using the y+ value as 350, the first layer thickness can be calculated as 1.84×10^{-3} m. The inflation layer is kept around wall and valve with the growth rate of 1.2 is kept in wall. The total thickness of inflation layer is kept enough to capture boundary layer phenomenon. The mesh independence study was performed with convergence criteria of 1% in only one model by taking head loss as parameter.

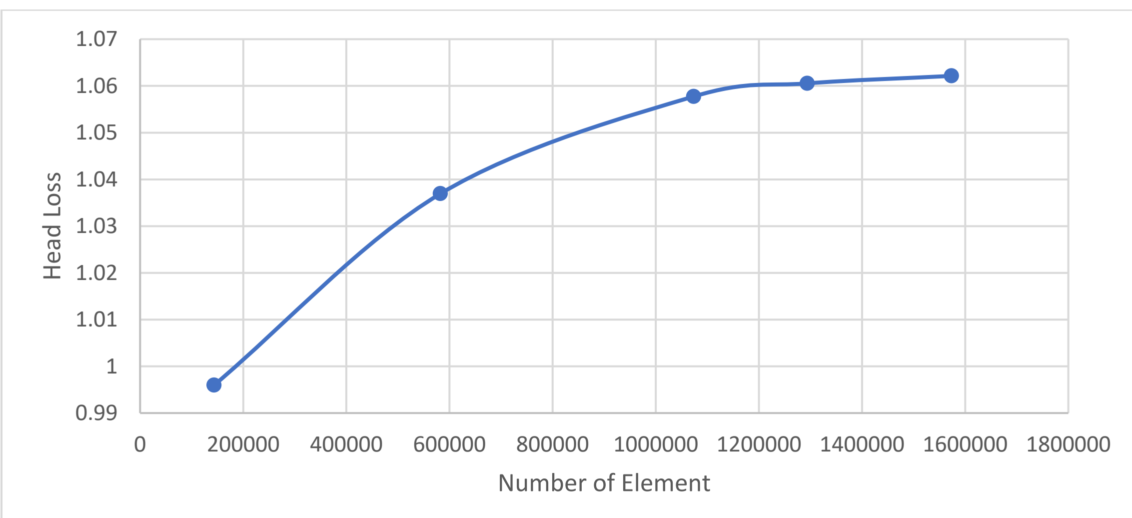

Figure 5 Mesh Independence Test

So, 1073673 numbers of elements were selected for the first model. On the other models, the method producing this amount of element was selected that includes face sizing of valve at 40 mm and body sizing as 200 mm. The body is divided into upstream body, valve body and downstream body. The upstream body and downstream body which are cylindrical are meshed with hexahedral mesh and valve body which is complex geometry is meshed with tetrahedral mesh.

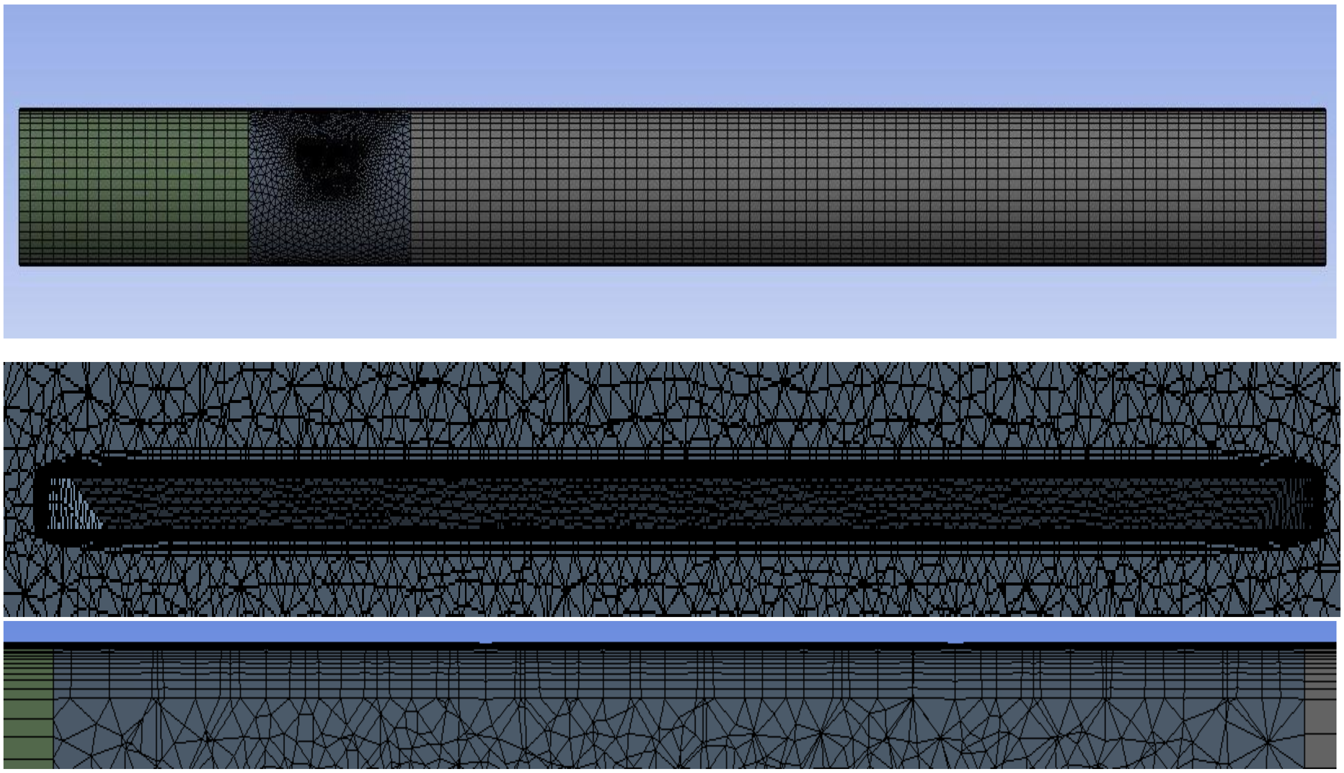

Figure 6 Mesh and Inflation layer in disc and pipe

The maximum skewness is 89.94% which is acceptable for the simulation.

Table 5: Setup and Simulation of CFD

Analysis	Steady State Analysis
Turbulence Model	SST k-w
Phase	Single Phase, Fluid
Material	Water (H2O-l)
Methods	Coupled
Residual Target	1e-5
Maximum Number of Iterations	10,000
Initialization	Hybrid

Boundary Condition

The pressure inlet mass flow outlet boundary condition is used [5]. The inlet pressure is corresponding to head i.e. 169 m and outlet is mass flow outlet derived from designed flow rate of turbine. The valve and pipes are considered as no slip wall with wall roughness 0.0015 mm.

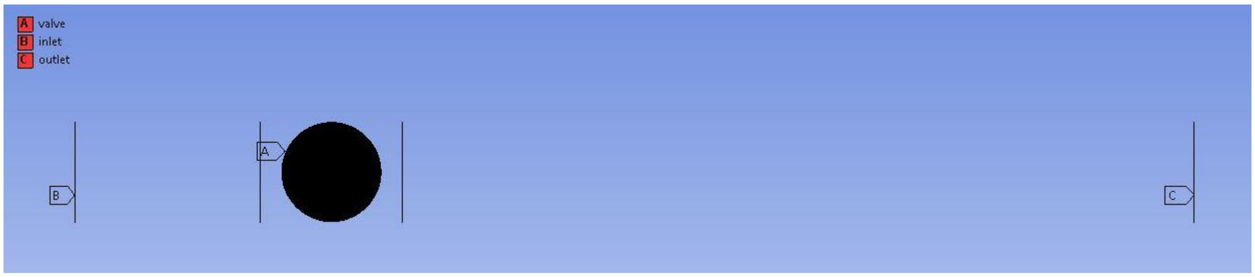

Figure 7 Named selection for Boundary Condition

Table 6: Boundary Condition for CFD Simulation

Boundary	Type	Value
Inlet	Pressure inlet	Pressure: 1,655,568.954 Pa
Outlet	Mass flow outlet	Flow rate: 63,500 kg/s
Pipe wall, valve	Wall	Roughness: 0.0015 mm

Pressure and Velocity Distribution

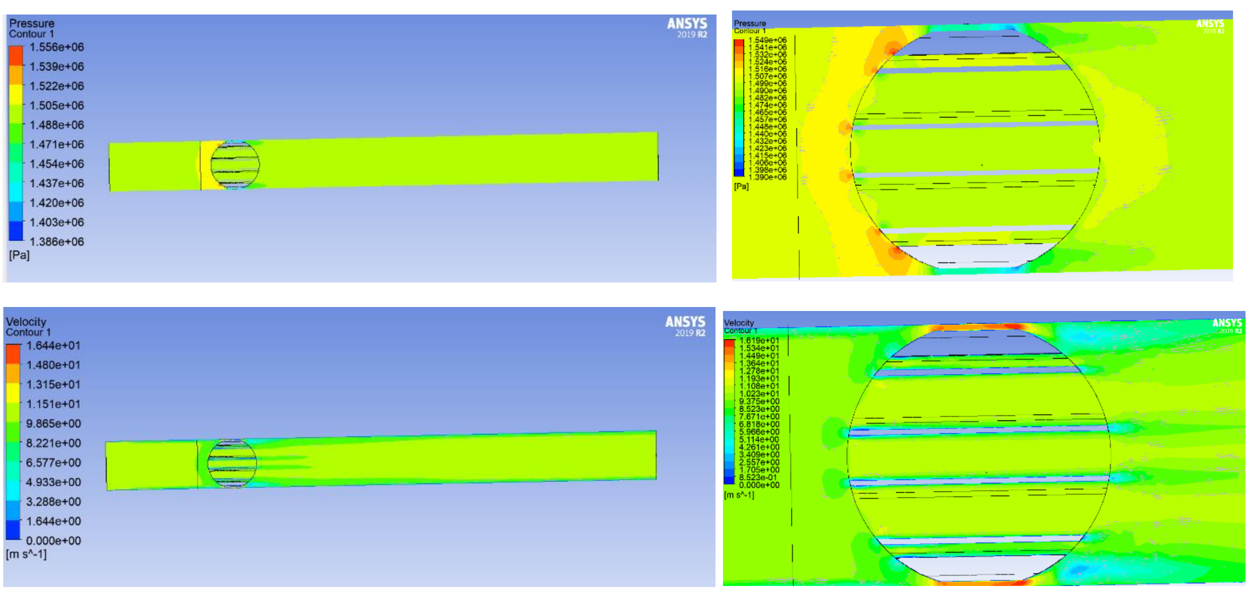

Figure 8 Pressure (Top) and Velocity (bottom) distribution in midplane

In the figure 8, the effect of rib can be observed clearly. Low velocity region is formed in front and back of ribs. In the wall side of valve, the velocity is high and it is due to small flow area. The pressure is maximum just in front of ribs and its is constant in downstream region.

Head Loss Calculation

Head loss is calculated by the formula [6]:

$$h_L = \{\frac{P_1}{\rho g} + \frac{V_1^2}{2g}\} - \{\frac{P_2}{\rho g} + \frac{V_2^2}{2g}\}$$

Where,

P_1 = pressure at inlet
P_2 = pressure at outlet
V_1 = velocity at inlet
V_2 = velocity at outlet

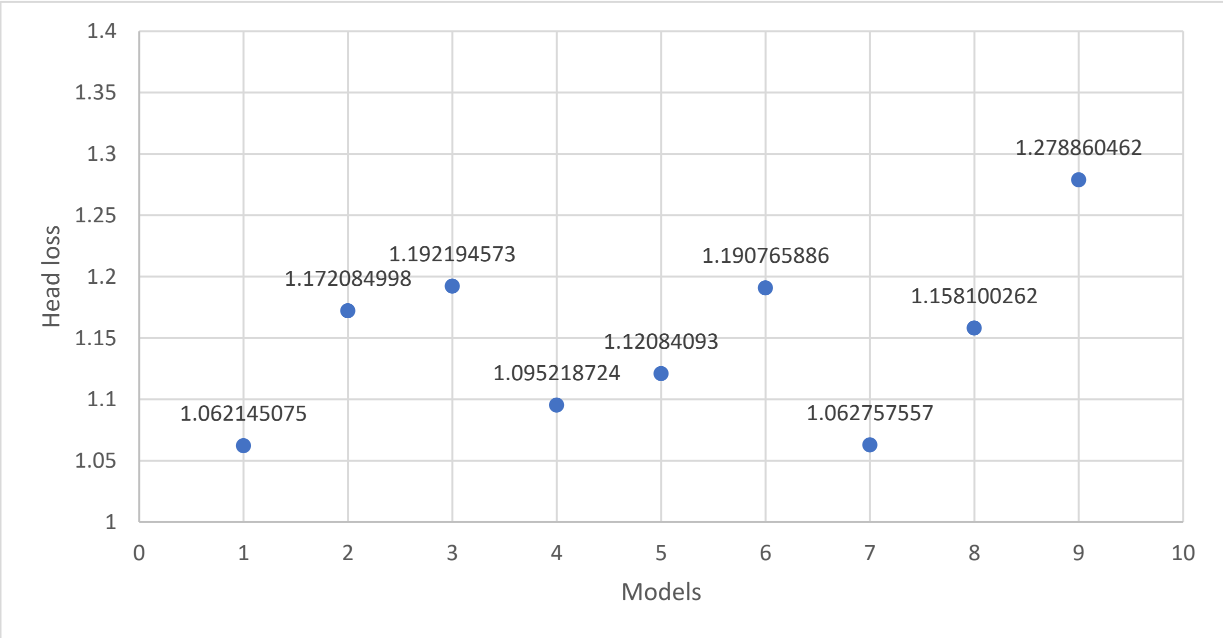

Figure 9 Comparison of Head Losses

It is seen that the parameter with significant effect in head loss is rib thickness and disc thickness is the parameter with least effect. The rib height somewhat affects the head loss. The valve disc is elliptical in nature, so it streamlines the flow. But, the shape of rib isn't streamlined and hence results in maximum head loss. The height of the rib increases the front cross section area and hence increasing the head loss.

CONCLUSION

Numerical simulation was performed for the optimization of Butterfly valve parameters for 169 m head and 63.5 m^3/s flow rate hydropower. All the models prepared for the simulation were structurally sound both from yield and total deformation criterion. The optimized model was then selected on the basis of minimum head loss. The model with disc thickness 173 mm, rib height 670 mm and rib thickness 50 mm was selected as final design. The maximum stress for this model was 262.91, maximum total deformation was 1.552 and head loss was 1.062 m.

REFERENCES

[1] Kacker, R. N., Lagergren, E. S., & Filliben, J. J. (1991). Taguchi's orthogonal arrays are classical designs of experiments. *Journal of research of the National Institute of Standards and Technology*, *96*(5), 577.

[2] SAMBIREDDY, E., & LAKSHMAIAH, V. (2017). Structural Design and FEM Analysis of Large Butterfly Valve.

[3] Huang, C., & Kim, R. H. (1996). Three-dimensional analysis of partially open butterfly valve flows.

[4] Menter, F. (1993, July). Zonal two equation kw turbulence models for aerodynamic flows. In *23rd fluid dynamics, plasmadynamics, and lasers conference* (p. 2906).

[5] Koirala, R., Chitrakar, S., Neopane, H. P., Chhetri, B., & Thapa, B. (2017). Computational Design of Bifurcation: A Case Study of Darundi Khola Hydropower Project. *International Journal of Fluid Machinery and Systems*, *10*(1), 1-8.

[6] Aguirre, C. A., & Camacho, R. R. (2014). Head losses analysis in symmetrical trifurcations of penstocks—High pressure pipeline systems CFD. *CamachoInstituto de Engenharia Mecânica, Universidade federal de Itajubá*.