

Weight Optimization of Large Size Butterfly Valve by Using Finite Element Analysis

Rohan Patil¹, Manzooraahamad Mirza², Sandesh Jadhav³, Suresh Sawant⁴, Himanshu Gaikwad⁵,
Akshay Kole⁶

Student, Department of Mechanical Engineering, RIT, Islampur, Maharashtra, India^{1,5,6}

Plant Head, Unique Hyrotech, Palus, Maharashtra, India³

Assistant Professor, Department of Mechanical Engineering, RIT, Islampur, Maharashtra, India^{2,4}

ABSTRACT:The objective of this study is to analyse stresses induced in a butterfly valve using the finite element method. Valves are a flow controlling device, which is commonly used in industrial applications for controlling internal of both compressible as well as Incompressible fluids. These valves are used for safety purposes, maintenance, shutoff and flow regulation as well as pressure regulation. Butterfly valve acts as a regulator in controlling the flow of the fluid. Butterfly valve consists of three main parts i.e. body, disc, and shaft. The primary objective of this work is to analyse stresses induced in the disc and valve body by FEM analysis. Pressure analysis for disc and body test considered by using standard IS 13095. The valve body and the plate are produced using Cast Iron GCI FG -260, GCI FG -300, GCI FG -350, GCI FG -400. The 3D displaying is to be performed by utilizing strong works programming for the butterfly valve. Additionally, by utilizing the ANSYS software, stress and displacement FEM study of the Butterfly valve is performed. The outcomes show stresses and distortion are inside the utmost. The 3D modelling to be performed for butterfly valve by using solid works software. Additionally, the stress & displacement FEA analysis of butterfly valve to be performed by using ANSYS tool to estimate the weight optimization results.

KEYWORDS:Butterfly valve, Body, Disc, FEM analysis, Weight optimization.

I. INTRODUCTION

Hydropower is considered as a standout amongst the most sensible and non-dirtying wellsprings of vitality. The power age from the water is named as Hydroelectricity. Hydroelectricity implies power produced by hydropower or from the utilization of the gravitational power of falling water or streaming water. A standout amongst the most widely recognized types of power age. since this type of vitality neither delivers direct waste issue nor it is exposed to depletion.

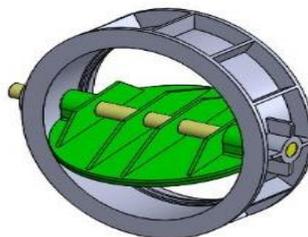


Fig.1.Butterfly valves

Presently multi day's increasingly more hydro electrical power plants being arrangement and revamped. Yet at the same time, the structure and improvement of hydro-electrical power plants depend on the customary strategies. In this manner, there is a colossal extent of use of cutting edge's procedures like limited component technique (FEM) for accomplishing most extreme conceivable enhancement. In any power plant valves for various objects are generally required. Typically, it is a stopped valve just before the turbine. Along these lines, the turbine might be discharged without exhausting the pole or penstock.

These particular Butterfly valves are introduced to secure hydro-electric establishments against the likelihood of penstock burst. Valves are intended to close against turbine runaway during a crisis condition. These valves go about as



secluding gadgets for the examination of penstock without dewatering whole head race burrow. One more significant application is for enormous water supply plans encouraged from repositories through long pipelines, the disappointment of which can cause sad floods. Valves by ideals of their structure and hearty development, close down naturally and momentarily in case of the downstream pipeline disappointment. These valves close during over speed condition happening because of turbine outing or runaway condition. Butterfly valves are regularly connected before low and medium head water turbines, for example heads up to 200 m. For high head power plants, the butterfly valve is now and again utilized as an end gadget in bay passages and on the other hand as crisis conclusion valves.

II. RELEVANCE

Butterfly valve are installed for protection of penstock deterioration. Valves are used to control the flow against turbine runaway during crisis condition. Valves in all respects generally utilized for a hydro-mechanical part which is utilized to control the progression of water under high heads. The butterfly valve is one of the kinds of shut-off gadgets most ordinarily utilized in hydropower stations and frameworks. Its utilization is favoured in view of their moderately minimal effort, smallness, lightweight, sensible water snugness and effortlessness of activity.

This research paper portrays the plan advancement of butterfly valves utilizing CAD software and ANSYS. To auxiliary structure, the butterfly valve for weight enhancement the von Mises stress and most extreme misshaping worth are discovered by utilizing ANSYS and think about stresses an incentive for existing threew sections streamlined manufactured plan for entryway and group of a butterfly valve. The examination of the butterfly valve is done to streamline the stresses and limit the weight utilizing ANSYS. The scientific model of improvement is set up initially, and the FEA is completed by utilizing the ANSYS software. In light of the investigation of an ideal outcome, the stress focuses on the valve has moved toward becoming assess, which gives a superior reference to update the valve.

III. BASIC DESIGN CRITERIA FOR BUTTERFLY VALVE COMPONENT

The design of valve components (valve disc, valve body & valve trunnion) reference consideration is taken from the Indian standard deals with the structural and hydraulic aspects of design, formulas are taken from various design books & various references from at her existing designs. The main characteristics of this butterfly valve are nominal diameter 1800 mm, nominal pressure 1.128 MPa, hydraulic test pressure Ph=1.962 MPa. The most commonly used material for the butterfly valve disc is Carbon steel, austenitic stainless steel.

A. Hoop Stress Calculation,

$$\sigma = \frac{P * D}{2t}$$

In above formula P, D, t and C are indicating Maximum working pressure (MPa), Maximum working pressure (MPa), Body shell thickness (mm), Minimum positive tolerance (MPa).

Table 1.Hoop Stress,

Sr. No.	Body shell (Internal Dia)	Shell Thickness (mm)	Pressure (MPa)	Hoop Stress (MPa)
1	GCI FG -260	36	0.8825985	22.1875
2	GCI FG -300	34	0.8825985	23.4927
3	GCI FG -350	32	0.8825985	24.961
4	GCI FG - 400	30	0.8825985	26.6251

B. Valve Body Design consideration

The valve bodies are generally fit into between two pipe flanges. The body and seat ring may be made in a single piece or separate in parts for the convenience of manufacture, customer requirement, easy to transport. The most well-known body designed drag and wafer type. The wafer type of valve is sandwiched between the pipe flanges and rib bolts encompass the body. Flow direction is important in the installation and use of a butterfly valve. Bodies may be casted or fabricated according to customer requirements and application purposes. Some bodies are coated with EPDM rubber coating, which increases the life of the valve. The design does not permit to use of through bolts, the flange has tapped



holes. The body should be stress relieved before any type of machining. The design should be sustaining maximum static head including pressure rise during working. Flange holes are bigger than the diameter of foundation bolts to allow for axial movement of the valve to avoid axial load on foundation.

C. Body shell thickness calculation

ASME Formula

$$t = \frac{P * R}{(S.E) - (0.6 * P)}$$

In above formula P indicates Design pressure in MPa, R indicates Internal Radius of shell in mm, S indicates Maximum Allowable stress Value in kg/mm², E indicates joint efficiency will be taken as 1.

Table 2.Body shell thickness

Grade	Ultimate Tensile Stress (MPa)	Yield Stress (MPa)	Allowable Stress(Kg/Mm ²)	T Provided (mm)
GCI FG -260	260	169	52	36.3606
GCI FG -300	300	195	60	34.3125
GCI FG -350	350	228	70	32.4107
GCI FG -400	400	260	80	30.9844

Table 2. Shows the results from the ASME formula. Four material having different values of thickness depends upon ultimate tensile stress, yield stress, and allowable stress values. There are a number of formulas for shell thickness calculations, but we use standard ASME formula.

D. Valve Disc Design consideration

The design of all types of discs should be such as to offer minimum head loss in the open position and to sustain the full differential pressure across the closed valve. The disc should be tight shutoff during testing or leak zero leaks is observed. The disc should be stress relieved before any type of machining. The shape of disc should not disturb the flow, and also it operates easily. The disc may be split into parts for ease of manufacture, transport, and erection. The disc should be stress relieved before taking up for machining operation. The angular travel of disc should be nearly 90⁰ from open to shut-off position. By using the following formula, we calculate the disc thickness of the disc. Assume a disc as a simply supported circular plate.

$$t = \sqrt{\left\{ \frac{3.W}{8.\pi.M.f} \left((3.M + 1) \times \left(1 - \frac{4 \times r^2}{D^2} \right) \right) \right\}}$$

In above formula ‘t’ indicates disc thickness in mm, ‘W’ indicates total load acting on disc, ‘M’ indicates reciprocal of passion’s ratio, ‘f’ indicates maximum allowable working stress in MPa.

Table 3. Disc Thickness

Sr. no	Material	Maximum Allowable Working Stress (MPa)	Calculated Disc Thickness (mm)
1	GCI FG -260	169	67.60
2	GCI FG -300	195	63.0
3	GCI FG -350	228	58.27
4	GCI FG -400	260	54.50

Table 3 shows the values which are calculated by disc thickness formula. by using different type of material different values are resulted. The Material and Maximum Allowable Working Stress values are given from standard. During disc cad model creation, we only focused on rib shape and thickness. Material properties like Density, Maximum Allowable Working Stress, Youngs modulus are all derived from Indian standard. Gray iron is a kind of cast iron which has graphite microstructure.

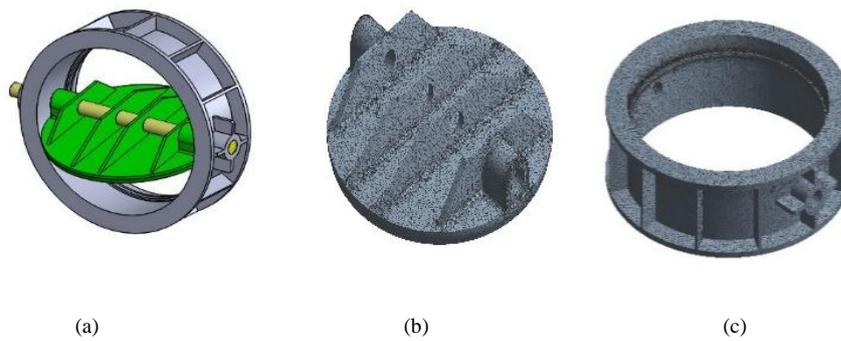


Fig.2.CAD Model Geometry(a) Assembled model (b) Meshing disc view (c) Meshing body view

Meshing is carried out with ANSYS software, a tetrahedral element was used for meshing the model with prism layer for modelling boundary layer with smooth transition as fig.2. After meshing we observed 197879Nodes ,1034194 coarse elements for Disc, 334403 nodal points and 101773 coarse elements for Body. Tetrahedral element mesh satisfies the quality like element quality, aspect ratio, jacobian ratio, warping factor etc. are with in limit.

IV. BOUNDARY CONDITION & LOADING FOR FEA ANALYSIS

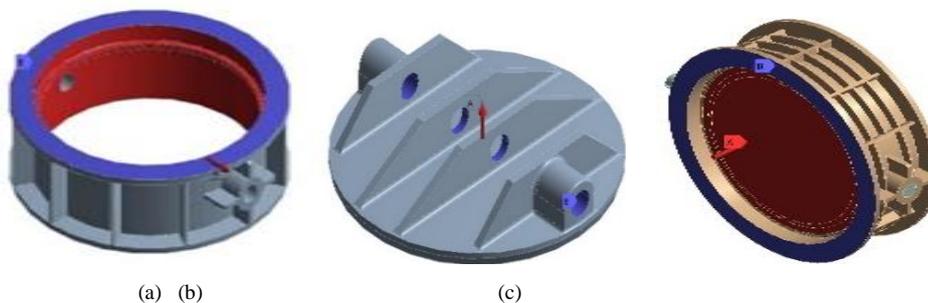


Fig. 3Boundary condition (a) Body(b) Disc(c) Assembly

Figure 3. shows the Boundary condition for the disc, body and assembly. The blue colour shows fixed support and red colour shows applied pressure. In case of disc, shaft is replaced by the fixed support to find out the strength of disc. in fig.3.b pressure is applied on the face of disc. Fig3.c shows the Boundary Condition for Assembly at closed position. Force is applied on disc and circular inner area of body shell. Disc is at fully closed position. Fixed support is applied to the body on both side flange.

V. SAMPLE SHAPES OF DISC & BODY

Finite element analysis is applied on 4 different sample shapes of disc. which have same boundary conditions with applied pressure. For weight optimization, we focus on stressed area by changing shape near to the excessive stress area. During disc shape creation, rib shape must be symmetric and it does not effect on body during disc rotation. In Second model the shape changed disc rib converted to semicircular shape. By changing the shape of rib check the possibility to weight reduction. Fig.4-disc sample 3 shows the empty triangular shape and disc sample 4 has addition of upper plate by reducing the rib thickness. During the creation of disc samples does not changes the boss shape or size. Disc should

be symmetric its rib dose not effect, when valve disc rotation. During body shape creation, nut bolt attachment does not disturb by the circular rib. During creation of body shape, it does not increase weight it but increases the strength.

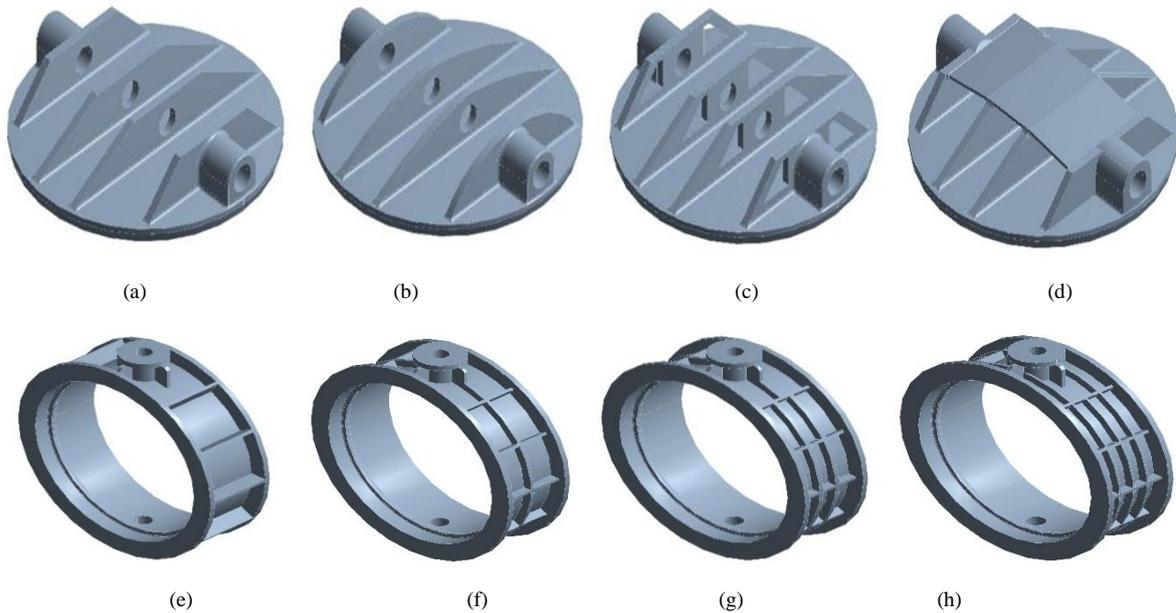


Fig. 4 Disc and body samples (a) Disc sample 1 (b) Disc sample 2 (c) Disc sample 3 (d) Disc sample 4 (e) Body sample 1 (f) Body sample 2 (g) Body sample 3 (h) Body sample 4

Body sample four shapes are shown in Fig. 4.e, f, g, h. First model shows only straight ribs between upper and lower flange. Body sample 'f' shows the one addition circular rib to decrease the central stress concentration. Body sample 'g' has two central circular ribs. First body sample model shows the rib width is same as to outer face of flange, but model sample f, g, h has half width of the flangewidth.

VI. FINITE ELEMENT ANALYSIS RESULT

Finite element analysis results are shown below. That test is divided into three parts body test, disc test, assembly test. During body test it is observed that sample 4 is showing very low stress as compared to sample 1. Due to increasing number of rib weight of body is also increased. Whereas as reduce that weight, we reduce the shell thickness up to optimal value.

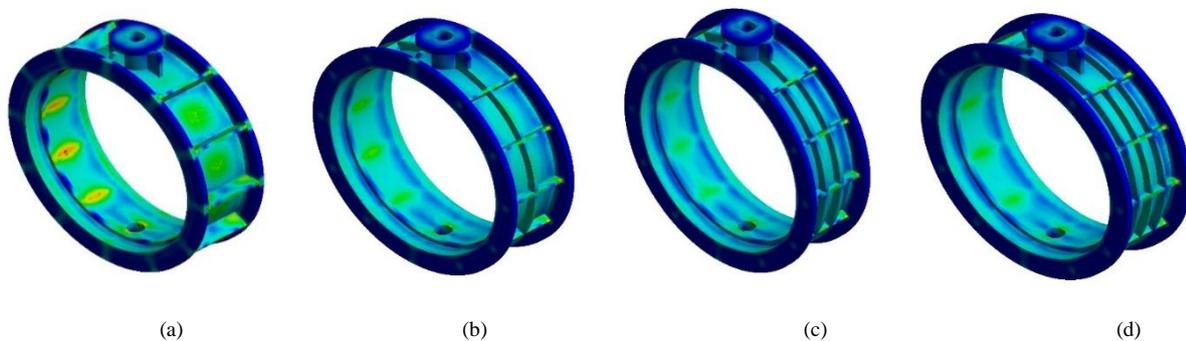


Fig. 5. ANSYS results for body (a) Body sample 1 (b) Body sample 2 (c) Body sample 3 (d) Body sample 4

Fig. 5 shows the sample a to dequivalent stresses are reduced. In Fig. 5.a first shape red colour area is occurring it decreased in Fig. 5.b and Fig. 5.c sample. The blue colour is observed in sample 4, that indicates the less stress is observed. stress sustain ability of sample Fig. 5.d is more than sample 1. Red and blue colour is reduced from sample Fig. 5.a to Fig. 5.d the Maximum stresses will occur at the central portion of inner side of the shell. by reducing

rib material Weight reduction is possible. During testing (ANSYS) to check the sample by changing shell thickness, changing material check. That same procedure is applied to all samples.

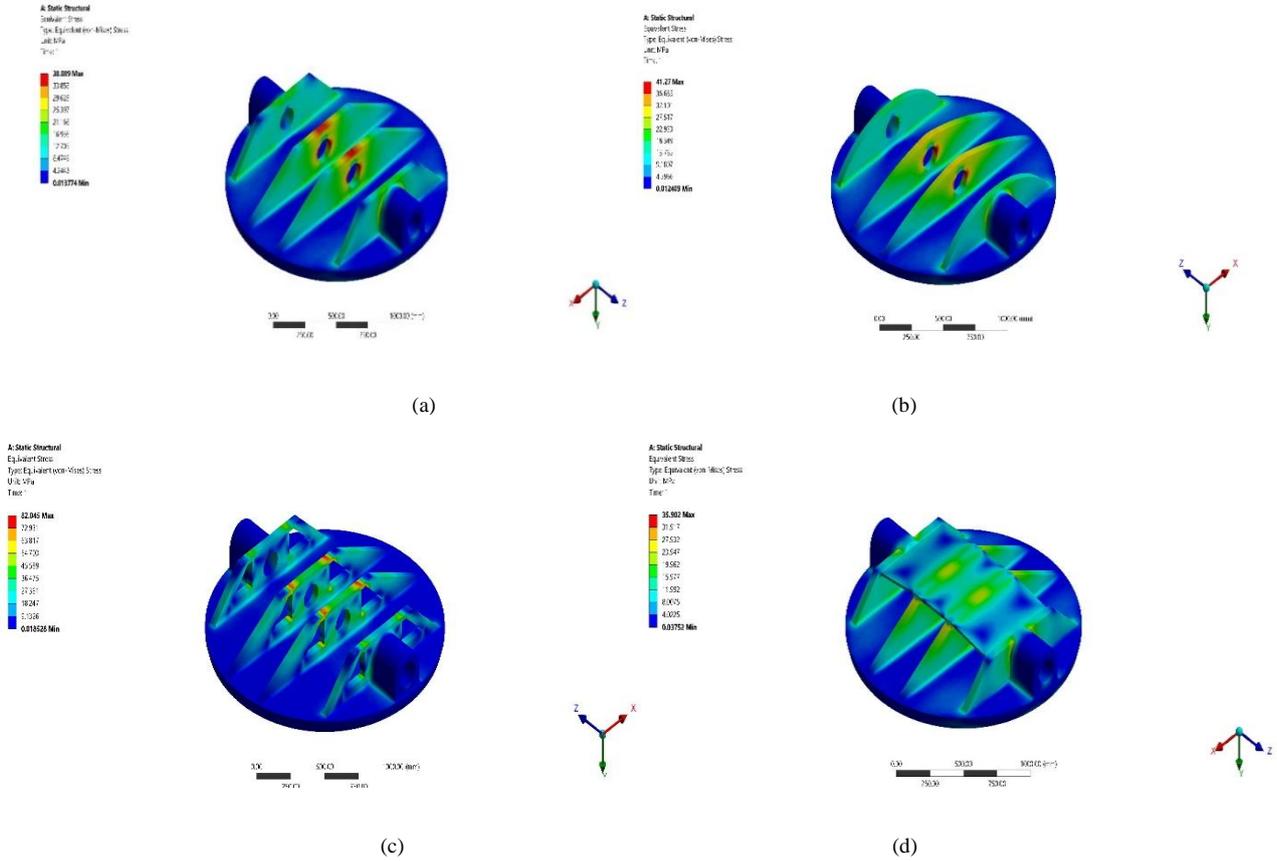


Fig. 6. ANSYS results for Disc (a) Disc sample 1 (b) Disc sample 2 (c) Disc sample 3 (d) Disc sample 4

Figures 6 shows the disc analysis results. Stress is increased in shape Fig. 6.b and c as compare to shape Fig. 6.a model. Stress is decreased in shape Fig. 6.d. when the force is applied on the disc the maximum stress is applied on the hole near the rib area. It is observed that by support the rib it increases the strength of disc. Due to addition of upper plate to the disc stress is reduced but weight is also increased. Weight reduces by decreasing the middle two rib thickness up to optimal solution.

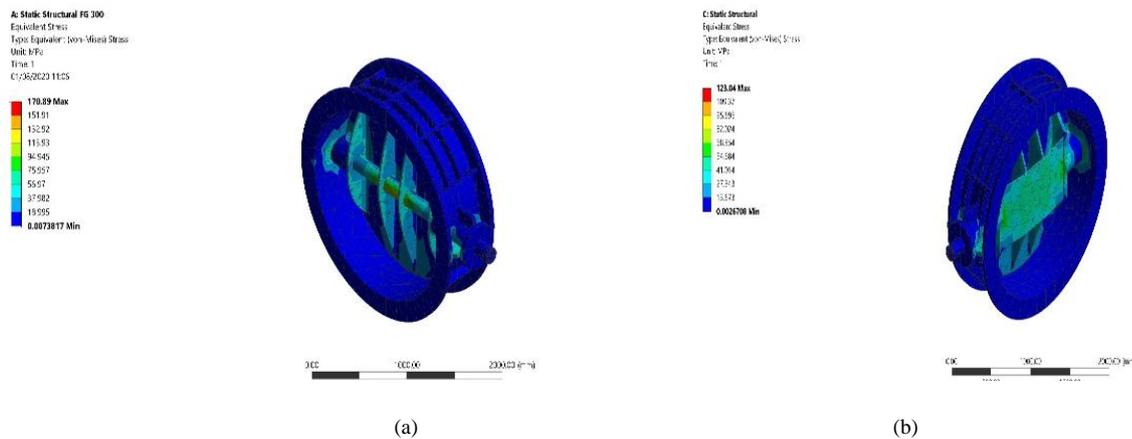


Fig. 7. Assembly Analysis results (a) Assembly (b) Assembly

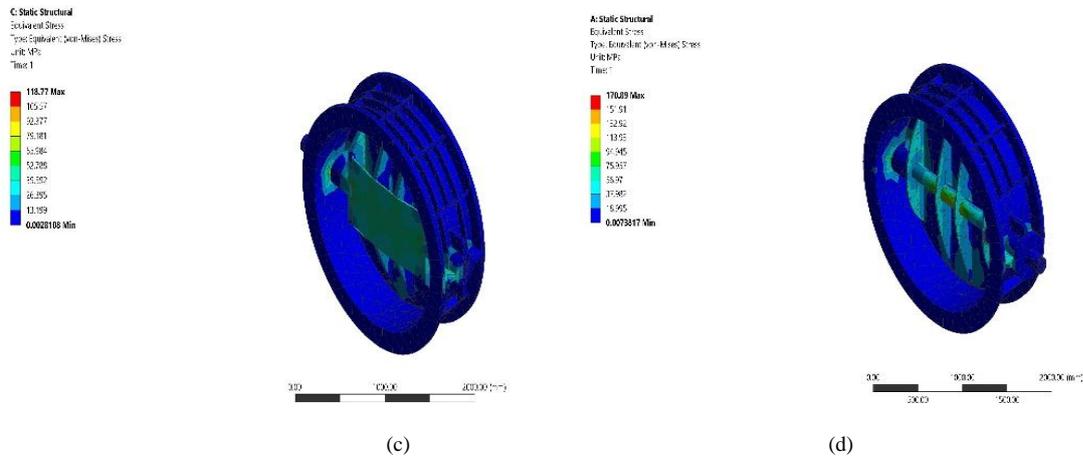


Fig. 7 Assembly Analysis results (c) Assembly (d) Assembly

VII. EXPERIMENTAL VALIDATION

Fig. 8. shows Experimental test which is conducted on the hydro test setup. A hydrostatic test is a one way in which pressure vessels such as pipe lines, gas cylinder, boilers, fuel tanks can be tested for strength and leak. During body test, it filled with liquid (water). After adding specific pressure visual leak detection is observed, if it is present. During body test 2 plates are attached to top and bottom side of the body by using stud and nut. Pressure is applied at the bottom side. We are applied the strain gauge to the outer shell centre. For body test purpose two-gaugerettes strain gauge method is used.

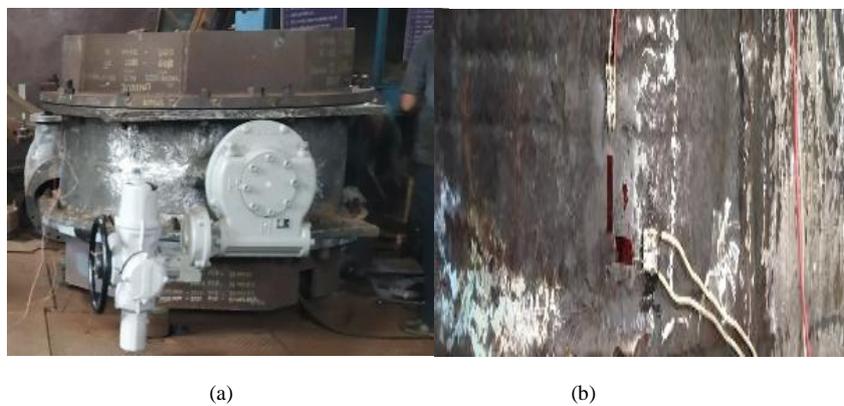


Fig. 8 Testing Setup (a) Body Test (b) Detail view of strain gauge mounting

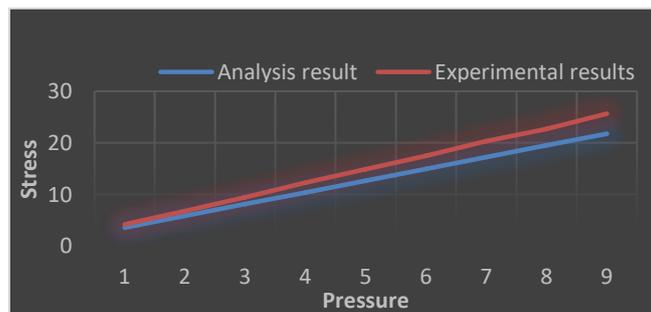


Fig. 9. Study Analytical vs experimental results

Initial it is observed that same analysis and experimental results are identical which shown in fig.9. Then after the pressure 6 to 9 kg/cm² graph difference will increase. This occurs because when we apply hydro pressure to the valve body, that pressure is controlled by small valves. When pressure exceed than 6 kg/cm² it quite difficult to off the valve at accurate pressure. Sometime more pressure is applied, that excessive applied pressure effects the large difference. The graph shows the 14 to 15% Error is observed between analysis and experimental results.

VIII. RESULT & DISCUSSION

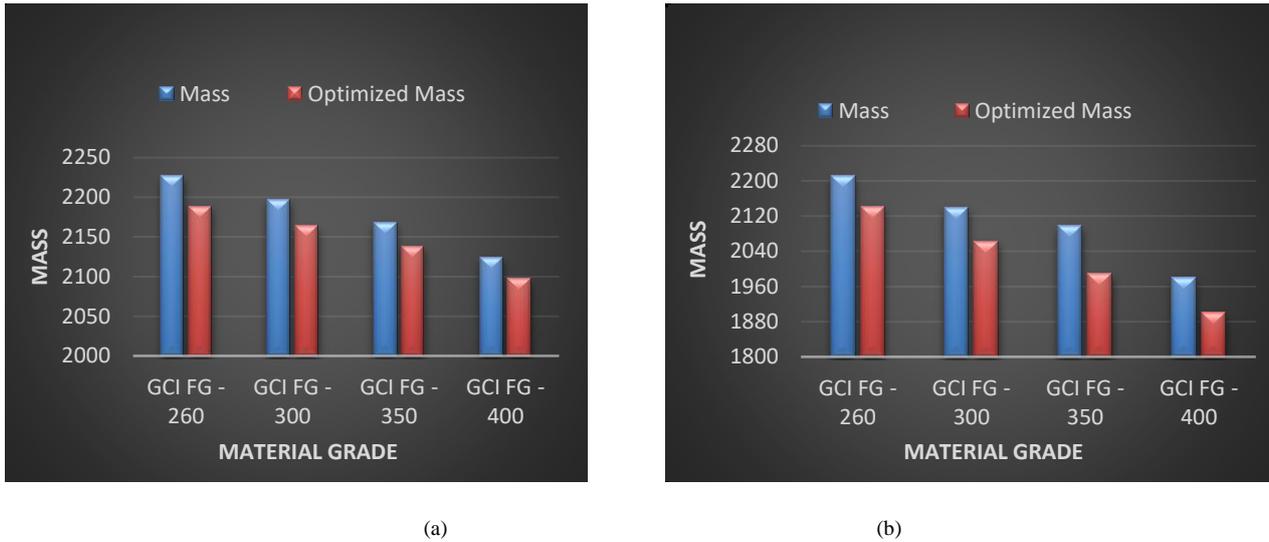


Fig. 10. Optimization Graph (a) Weight optimization comparison sheet for body (b) Weight optimization comparison sheet for disc

Figure 10. shows the graph material grade vs mass of the body. The Grades GCI FG 260,300,350 & 400 mass differences between initial body mass and optimized body mass are 37.9 kg,33.6 kg,29.3 kg,26.3 kg. By changing sample shape and material 128.8 kg weight can be reduced. That indicates the combination of sample 4 and GCI FG 400 material gives better results. Figure 10.b. shows the initial disc weight and optimized weight results. The graph shows it continuously reduced by changing the shape and material grade. The maximum weight reduction is observed at GCI FG 400 material. The weight difference 68.1 kg,76.8kg,77.3kg and 77.7kg are observed for GCI FG 260,300,350 & 400 material.

Table 4: Weight optimization comparison sheet for butterfly valve assembly of different material grade

Sr. No	Material Grade	Stress (Mpa)	Optimized Model Stress (Mpa)	Safety Factor (Minimum)	Optimized Model Safety Factor (Minimum)
1	GCI FG -260	170.74	144.67	1.4642	1.4087
2	GCI FG -300	174.97	132.78	1.4288	1.8803
3	GCI FG -350	176.97	131.25	1.417	1.9871
4	GCI FG - 400	187.14	128.83	1.3359	2.1132

Table 4 shows the assembly results for old and new sample model. It is observed that optimized model stress is less than initial and factor of safety is increased. We find that, the Von Mises Stress produced in the parts of Butterfly Valve due to applied pressure of 1.962 N/mm², are less than the yield strength of the material. Hence, we conclude that, design of butterfly valve for chosen material is safe. Material GCI FG 400 has maximum safety factor is observed as compare to other materials. We compared the initial design with the optimal design. Analysed that the weight reduction up to 39 percentage is possible in the modified design compared to the initial design.

IX. CONCLUSION

This research includes information about design specifications of 1800mm size of Butterfly Valve. Butterfly valve CAD model created by using solid works software. Results are found by changing rib shapes. In Butterfly Valve disc - maximum equivalent stresses occur near the boss. Final optimized model sample 4 of disc shows the factor of safety is more than previous base model. Stress produced in sample 4 is also less than initial model. The final optimized Disc model shows reduction 13.98 % of weight.

During body analysis maximum stress is located near to the central area of shell. Body optimized results are found by changing the rib shape, material. Final optimized model sample 4 of body shows greater factor of safety than initial model.

Sample 4 body has less stress observed than the initial model. Number of assembly model tests that results show the sample 4 disc and sample 4 body good results. Von Mises equivalent stress do not exceed than 260 N/mm², Actual observed stress 121.43 N/mm² which are less than allowable stress, it concludes that modified design becomes safe. The final optimized body model shows reduction 5.79 % of weight.

Maximum Von Mises stresses induced in the assembly is 123.87 MPa which is less than yield strength (228 MPa) at given pressure condition. After the study of experimental and analytical results it shows that the final optimized butterfly valve design is safe. From the above results it is observed that maximum von Mises stresses induced in the disc, body and assembly are less than yield strength at given pressure condition. Hence, we conclude that the Final optimized model of butterfly valve is safe.

REFERENCES

- [1] C. Hung and R.H. KIM, (1996), "Three-dimensional analysis of partly open butterfly valve flows", journal of fluids engineering, Trans.ASME,118 562-568.
- [2] M.J. Morris and J.C. Dutton (1991), The performance of two butterfly valve mounted in series, Journal of fluids engineering,113(3) 419-423.
- [3] Song, X., and Park, Y.C., 2007, "Numerical Analysis of Butterfly Valve - Prediction of Flow Coefficient and Hydrodynamic Torque Coefficient," In Proceedings of the World Congress on Engineering and Computer Science, pp. 759-763.
- [4] Park JuYeop, Chung Myungyoon. (2006), study on hydrodynamic torque of a butterfly valve, journal of fluids engineering;128(1);190-5.
- [5] Huang, C., and Kim, R.H., 1996, "Three-dimensional Analysis of Partially Open Butterfly Valve Flows," J. Fluids Engineering, 118(3), pp.562-568.
- [6] Ogawa k, Kimura T. (1995), Hydrodynamic characteristics of butterfly valve prediction of torque, ISA Transaction; 34:327-33.
- [7] T. Sarpkaya (1961), Torque and cavitation characteristics of butterfly valve, journal of applied Mechanics, Trans, ASME 511-518.
- [8] Song, X., Wang, L., and Park, Y., 2008, "Fluid and Structural Analysis of Large Butterfly Valve," AIP Conference Proceedings, 1052, pp.311-314.
- [9] FXue Guan 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.
- [10] Danbon F Sollic c. (2006), Aerodynamic torque of a butterfly valve – influence of an elbow on the time mean and instantaneous aerodynamic torque. journal of fluids engineering;128(1):190-5.