ANYALYSIS AND DESIGN OPTIMIZATION OF 8''- 600# GATE VALVE BODY USING FEA AND STRESS ANYLASIS

Pujari A.A[1], Joshi G.S[2]

[1]PG student , Department of Product Design and Development, D.K.T.E's Textile and Engineering Institute, Ichalkaranji , Shivaji University, Kolhapur, Maharashtra, India
[2]Assistant Professor, Department of Mechanical Engineering, D.K.T.E's Textile and Engineering Institute, Ichalkaranji, Maharashtra, India.

Abstract

A gate valve can be used for a wide variety of fluids and provides a tight seal when closed. Gate Valves are designed to suit a wide range of applications in Refineries, Petro-chemical Complexes, Fertilizer Plants, Power Generation Plants (Hydro - electric, Thermal and Nuclear) Steel Plants and Allied Industries. They are made from high quality Carbon Steel Castings and embody design features that contribute to strength and durability. Gate valves are used when a straight-line flow of fluid and minimum restriction is desired. Gate valves are so named because the part that either stops or allows flow of fluid through the valve acts somewhat like the opening or closing of a gate and is called, appropriately, the gate. The objective of this paper is to perform a literature review on optimization of various mechanical parts.

Keywords: *Gate Valve, Finite element method, Stress*

--- ***---

1. INTRODUCTION

A Gate Valve, or Sluice Valve, as it is sometimes known, is a valve that opens by lifting a round or rectangular gate/wedge out of the path of the fluid. The distinct feature of a gate valve is the sealing surfaces between the gate and seats are planar. The gate faces can form a wedge shape or they can be parallel. Gate valves are sometimes used for regulating flow, but many are not suited for that purpose, having been designed to be fully opened or closed. When fully open, the typical gate valve has no obstruction in the flow path, resulting in very low friction loss and when the gate valve is closed there are many obstructions in the flow path which in turn produces high frictional losses.

2. THEORY OF GATE VALVE

A gate valve is a linear motion valve used to start or stop fluid flow; however, it does not regulate or throttle flow. The name gate is derived from the appearance of the disk in the flow. Fig .1 illustrates a gate valve. The disk of a gate valve is completely removed from the flow stream when the valve is fully open. This characteristic offers virtually no resistance to flow when the valve is open. The basic parts as shown in fig 1 are as follows:-

1. Valve Bonnet: The cover for the opening in the valve body is the bonnet. In some designs, the body itself is split into two sections that bolt together. Like valve bodies, bonnets vary in design. Some bonnets function simply as valve covers, while others support valve internals and accessories such as the stem, disk, and actuator.

2. Valve Trim: The internal elements of a valve are collectively referred to as a valve's trim. The trim typically includes a disk, seat, stem, and sleeves needed to guide the stem. A valve's performance is determined by the disk and seat interface and the relation of the disk position to the seat.

3. Disk and Seat: For a valve having a bonnet, the disk is the third primary principal pressure boundary. The seat or seal rings provide the seating surface for the disk. In some designs, the body is machined to serve as the seating surface and seal rings are not used.

4. Stem: The stem, which connects the actuator and disk, is responsible for positioning the disk. Stems are typically forged and connected to the disk by threaded or welded joints.

5. Valve Actuator: The actuator operates the stem and disk assembly. An actuator may be a manually operated hand wheel, manual lever, motor operator, solenoid operator, pneumatic operator, or hydraulic ram

6. Valve Packing: Most valves use some form of packing to prevent leakage from the space between the stem and the bonnet. Packing is commonly a fibrous material (such as flax) or another compound (such as Teflon) that forms a seal between the internal parts of a valve and the outside where the stem extends through the body.

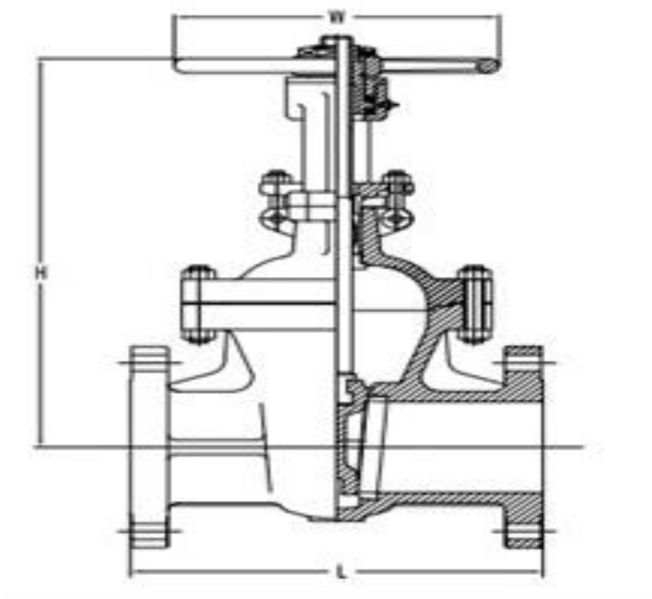

Fig- 1: Gate Valve Diagram

3. LITERATURE WORK

The aim of the literature review was to get the detail about the approach of methodology adopted in carrying out analysis and design optimization of 8”-600 class gate valve.
E.S. Barboza Neto, et al. [1] has investigates the behavior of low linear density polyethylene (LLDPE) and 5 wt.% of high density polyethylene (HDPE) They made prototypes of pressure vessel used for the storage of the compressed natural gas (CNG).The described method for theoretical determination of minimum thickness and ratio of burst pressure for distinct size liners. FEA simulations of the composite pressure vessel were conducted using ABAQUS/CAE 6.8, considering linear elastic behavior of the polymeric liner and the carbon/epoxy laminate. The elasto-plastic model used for the polymeric blend liner material showed agreement with the experimental pressure tests and the ASME procedure.

Xue-Guan SONG et al. [2] studied the mechanical and chemical properties of CF8M through experiments. An application of CF8M in valve body was analyzed by using finite element method (FEM) to evaluate the structural safety. They used the computer technique to overcome the difficulty of non transparency difficult to investigate details of the flow inside a ball valve, because it was not visible. An optimization containing several variables based on the response surface method (RSM) was conducted to find the optimum dimension of the valve. The results show that using FEA process can save valve mass as well as the computational expense effectively.

Moustabchir et al. [3] had presented experimental & numerical study analysis of stress strain state of pressurized cylindrical shell with external defect. Firstly A cylindrical shell with longitudinal external defects subjected by internal pressure was investigated. Model specimens of a cylindrical ferrule shape with two tori spherical lids were manufactured of steel P264GH.

Yong Zhang et al. [4] had worked on new method combining genetic algorithm with finite element simulation to obtain the optimal forming parameters of hydro forming. They worked on the ball valve shell as it is the key component in assembly contributing the weight. Paper describes the tube hydro forming (THF) process which is used mainly fluid pressure and tube material to produce various shaped parts. The FE simulation is also carried out.

B. Prabu et al. [5] has worked on the buckling failure of the cylindrical pressure vessel. Paper describes the Eigen buckling analysis which predicts the theoretical buckling strength of an ideal linear elastic structure. Along with Eigen buckling FE analysis for non linear problems also described. Results obtained are validated by comparing it with buckling pressure given in the data book.

L.N. Wankhade et al. [6] In this paper, the wok is carried out to measure the stress and temperature distribution on the top surface of the piston. In I.C. Engine piston is most complex and important part therefore for smooth running of vehicle piston should be in proper working condition. Pistons fail mainly due to mechanical stresses and thermal stresses. Analysis of piston is done with boundary conditions. The CAD model is created using CATIA V5 tool. CAD model is imported into the Hyper Mesh for geometry cleaning and meshing purpose. The FEA is performed by using RADIOSS.

Prabhala et al. [7] It has investigate that the mileage of the automobile also depends on the weight of the automobile. And the major weight is engine. As the engine is the assembly of many components, we will take the particular component and optimization of weight is done i.e. with respective to its function. In this project they take the aluminum alloy 1060 alloy and cast alloy steel. The components are designed by using pro-E and analysis is done by cosmos. Design of engine crankshaft, connecting rod and piston assembly is analyzed by using the standard forces acting on the piston. By using steady state and modal analysis measure at different connection, observing the above analysis results Aluminum is having very less weight comparing to the steel so that we can conclude that modified assembly is having more mechanical efficiency and also they can reduce the cost of the product and production .

Deokar Vinayak Hindurao, D.S.Chavan et al. [8], author in their paper named Optimization of 16” Plug Valve Body Using FEA And Experimental Stress Analysis Method, discusses FEA analysis of Plug valve body followed by Experimental stress analysis using strain gauge method for weight optimization. New optimized models were prepared on the basis of validation of the results obtained from stress analysis procedure. The weight reduction is done by changing the wall and rib thickness.

Dr. K.H.Jatkar, Sunil S. Dhanwe et al. [9], the objective of paper named Finite Element Analysis of Gate Valve is to perform a stress analysis of the critical component of Gate Valve. The critical components in the Gate Valve are Body, Gate Stem and slab gate. This paper comprises Finite element analysis of Gate Valve. A model of each element of Gate Valve is developed in CATIA V5R17, and analyzed in ANSYS 11. Gate valve stress analysis is done by FEM using ANSYS 11 and valid action is supported by stress analysis using classical theory of mechanics.

Nermina Zaimović-Uzunovi ć, Ejub Ajan, Samir Lemeš et al. [10], studied in the paper Weight optimization of the butterfly valve housing. This paper gives basic methodology of butterfly–valve design by using CAD technologies and FEM. Main purpose is weight optimization of the valve housing body.

Pradnyawant .K. Parase, Prof. Laukik B. Raut et al. [11], in the paper named Weight optimization of 12"-150 Class Plug valve Casting Body by Finite Element Analysis , a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure.

Silva ,S .P., Pereira , R.F.P.,Abreu, G. A., Panzera, T. H., Brandão et al. [12], This research focus on the investigation of new cutting tools applied on the manufacturing process of safety systems for the automotive industry. The sleeve machining sector of pinion & valve assembly for steering system was investigated to compare the old tools with the new ones with Penta geometry considering the lean manufacturing concept. In 2009 the data showed a percentage increase of 185% for the valve sector productivity. It was verified that the sleeve machining time were reduced providing a 100% of productivity demanded. In addition, despite of the investments in new tools, there was a cost reduction for each sleeve, consequently increasing the annual gain for the installed capacity.

J.S. Rao, Bhaskar Kishore, Vasantha Kumar et al. [13] , In this paper weight removal can be optimized to obtain maximum reduction without compromising the structural integrity. This procedure is illustrated in this paper by using two optimization codes, Altair OptiStruct for linear structures and Altair HyperStudy for nonlinear structures using Ansys platform as the main solver. Using this procedure nearly 10% weight reduction is achieved.

Mona Golbabaei Asl, Rouhollah Torabi, S. Ahmad Nourbakhsh, Karo Sedighiani et al. [14] Author worked in paper Failure Detection and Optimization of a Centrifugal-pump Volute Casing. This paper presents fatigue life prediction both for a real failed volute casing and an optimized one. Volute-casing optimization can be achieved once its mechanical behavior is known from finite element results. The results are revealed by structural analysis of casing under certain standard test conditions.

4. CONCLUSION

From the papers referred above many conclusions can be drawn. These are summarized below:

1. Analyzing the results, the ideal thickness of the actual liner to withstand a maximum pressure 2.0 – 2.2 MPa was found to lie within the 15–16 mm range. In order to account for manufacturing tolerances, a 15.3 mm nominal thickness was chosen for the liner. The hydrostatic pressure limit of the actual liner was 2.0 MPa and the characteristic fracture behavior changed from brittle to ductile.
2. Experimentation concluded that 1) 16.67% weight is reduced from the initial design 2) The ASTM A296 CF8M is very suitable to ball valve, as it enhance general corrosion resistance and to provide greater strength at room and elevated temperature. They suggested including the thermal analysis in optimization in the future work.
3. Results show that a FEA and Strain gauge result matches with each other and are acceptable.
4. The results show that the proposed method seems to be able to find the global solutions of the forming parameters for thin-walled valve shell.
5. This paper concludes that when amplitude of imperfections increases the buckling pressure decreases and as the λ of the inward dimple increases, the buckling strength decreases.
6. By using the simple concepts of FEA we were able to find critical areas of failure of model. The piston experiences maximum stress in the region where the combustion of the fuel takes place, i.e. at the piston head. Topology optimization using Altair's optimization software OptiStruct found to be very useful for generating new concept designs in less time.
7. Aluminium is having very less weight comparing to the steel so that we can conclude that modified assembly is having more mechanical efficiency and also we can reduce the cost of the product and production (Aluminium products are manufactured in cold chamber castings, steel products are manufactured in hot chamber castings).
8. The results clearly shows the maximum weight reduction is 24.86 kg (5.26%) weight of original weight while keeping maximum stress level up to 168.6 N/mm^2 which is safe for the applied load.
9. The stress values obtained by classical theory of mechanics & stress values obtained by Finite element method (FEM) are approximately same.
10. Optimization of the complex shape constructions, such as the housing of the butterfly valve can be successfully performed using CAE techniques. In this case, the valve housing (D = 600 mm and p = 16 bar) was optimized by reduction of the weight.
11. Results of decreasing the wall thickness and increasing the neck size are better than only reducing the wall thickness.

12. The Penta combined tool allowed to work with small values of cutting parameter, enhancing the tool life; The Penta combined tool contributed to the reduction of machining time , contributing to the increase of productivity capacity
13. The shape optimization showed that the local strain can be reduced considerably by as much as 26%. This reduction will have significant influence on the life of the bladed disk. In case of weight optimization of the blades is considered. In the root region where there is a considerable material with less stress load distribution, several holes and cutouts in the shank region are used as design variables. The weight of the blade root region where the cutouts are made is taken as the objective function. The shank cut outs and the holes in the root are used as design variables. The root region could be optimized to reduce weight by as much as 10%.
14. Experimental fatigue life and S-N prediction are in reasonable agreement; nevertheless, S-N curve is conservative for higher cyclic load levels. Therefore, modification applied to the initial geometry of the volute casing enhances its mechanical reliability and working life from about 2000 hours up to 10000 hours.

REFERENCES

[1]. E.S. Barboza Neto , M. Chludzinski , P.B. Roese , J.S.O. Fonseca , S.C. Amico , C.A. Ferreira (2011),“ Experimental and numerical analysis of a LLDPE/HDPE liner for a composite pressure vessel”,*ELSEVIR. Polymer Testing* 30,pp. 693–700.

[2]. Xue-Guan SONG, Seung-Gyu KIM, Seok-Heum BAEK, Young-Chul PARK, (2009). “Structural optimization for ball valve made of CF8M stainless steel” *ELSEVIR.Trns.nonferrous Tet, Sec.China* 19. Pp.258-261.

[3]. H.Moustabchir, (2010),“Experimental and Numerical study of stress-strain of pressurized cylindrical shells with external defects”. *Equipe de Mecanique Materiaux et Systemes (EMMS)*, Department de Physique Faculte des sciences et Techniques Errachidia, University My Ismail, BP 509 Boutalamine, 52000 Errachidia, Morocco. Engineering Failure Analysis 17.pp.506-514.

[4]. Yong Zhanga,b, Shengdun Zhaoa, Zhiyuan Zhanga, (2008) .“Optimization for the forming process parameters of thin-walled valve shell”, School of mechanical engineering, Xi’an Jiao Tong University,Xi’an 710049, China. School of Mechanical and Electrical Engineering, Inner Mongolia Agriculture University, Inner Mongolia 010018, China. Thin Walled Structure 46.pp.371-379.

[5]. B. Prabu, N. Rathinam, R. Srinivasan, K.A.S. Naarayen, (2009). “Finite Element Analysis of Buckling of Thin Cylindrical Shell Subjected to Uniform External Pressure”. Department of Mechanical Engineering, Pondicherry Engineering College, Pondicherry-605014, India, *Journal of Solid Mechanics* Vol. 1, pp.148-158.

[6]. Dr. L.N.Wankhade, Vivek Zolekar (2013),” Finite Element Analysis and Optimization of I.C. Engine Piston Using RADIOSS and OptiStruct” ISSN 2141-2383.pp 1-8.

[7]. Sasi Kiran Prabhala, K. Sunil Ratna Kumar (2012) “Design and weight optimization of I c engine”,E-ISSN 2249–8974.pp 56-58.

[8]. Mona Golbabaei Asl, Rouhollah Torabi, S.Ahmad Nourbakhsh (2009), “Experimental and FEM failure analysis and optimization of a centrifugal-pump volute casing”,Hydraulic Machinary Research Institute, Mechanical Engineering Department, Faculty of Engineering, University of Tehran, P. O. Box: 11155-4563, Tehran, Iran. EBARA Pumps Machinery Company (EPMC), P. O. Box: 15875-7653, Tehran, Iran. Engineering Failure Analysis.1996-2003.

[9]. Chardrakant S.Desai, Abel, (2005)“Introduction to the finite element method.”.*CBS Publishers and Distributers, Edition 16*.pp 38-128.

[10]. Nermina Zaimović-Uzunovi ć, Ejub Ajan, Samir emeš, (2006).” Weight optimization of the butterfly valve housing” , 10th International Research/Expert Conference ”Trends in the Development of Machinery and Associated Technology” TMT 2006, Barcelona-Lloret de Mar,Spain, 11-15 September, 2006 .

[11]. Pradnyawant .K. Parase, Prof. Laukik B. Raut, (2014). “Weight optimization of 12"-150 Class Plug valve Casting Body by Finite Element Analysis”, *International Journal of Engineering Research & Technology (IJERT)*, ISSN: 2278-0181, Vol. 3 Issue 12, December-2014.

[12]. Silva, S .P., Pereira , R.F.P.,Abreu, G. A., Panzera, T. H., Brandão, (2010). “Machining Optimization of Sleeve and Valves of Hydraulic Steering System”, *Proceedings of the World Congress on Engineering 2010*, Vol III WCE 2010,June 30 -July 2,2010,London,U.K.

[13]. J.S. Rao, Bhaskar Kishore, Vasantha, (2011). “Weight optimization of turbine blades”, *Copyright Altair Engineering*, Inc., 2011, pp 1-19.

[14]. Mona Golbabaei Asl, Rouhollah Torabi, S. Ahmad Nourbakhsh, Karo Sedighiani, (2009). “Failure Detection and Optimization of a Centrifugal-pump Volute Casing”, *Proceedings of the SEM Annual Conference*, June 1-4, Albuquerque New Mexico USA.

BIOGRAPHIES

Amit A. Pujari obtained his Bachelor o f Techn Engineering degree in Mechanical Engg. n from Shivaji University, Kolhapur, India. And He is pursuing his Master’s d degree in product Design and Development from D.K.T.E’s Textile and Engineering institute, Ichalkaranji , Shivaji University, Kolhapur, Maharashtra, India. He is doing his project work on Design optimization of Gate Valve body using FEA and Stress analysis.He is working as a lecturer in Dr. Bapuji Salunkhe Institute of Engineering and Technology, Kolhapur.

Joshi G. S. completed his Masters degree I in Power Engg. With honors in Mechanical yea Engg. from Walchand College of Engg. , S Sangali, 2002. He is pursuing his PhD in Power Engg. from Spacific University. He is presently working as Assistant Professor from last 15 years in Mechanical Engineering department at D.K.T.E. College, Ichalkaranji, Maharashtra, India