Analysis and design optimization of 8"- 600# gate valve body using FEA and stress analysis.

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

[1] Professor, Department of Mechanical Engineering, D.K.T.E's Textile and Engineering Institute, Ichalkaranji, Maharashtra, India.

[2]PG student , Department of Product Design and Development, D.K.T.E's Textile and Engineering Institute, Ichalkaranji , Shivaji University, Kolhapur, 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, 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 analysis and design optimization on gate valve body.

***Key Words*: *Gate Valve, Finite element method, Stress Analysis method*, strain *indicator etc.*

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.
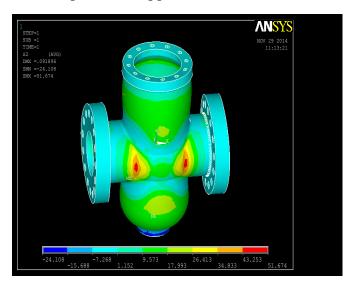
2. FINITE ELEMENT ANALYSIS

Simple mathematical model can be solved analytically, but more complex model requires use of numerical methods. FEA is one of the numerical methods used to solve complex mathematical problem. The entire solution domain must be discretized into simply shaped sub domain called as elements. ANSYS software is used for the analysis of the valve body, which is based on the FEA method.

Steps in finite elements analysis:-

1. 3D modeling of Valve Body 2. Meshing of the 3D valve body model 3. Material properties assigned 4. Loads and boundary conditions 5. Results and their physical interpretation

2.1 FEA results for 10.34 MPa and 15.51 MPa internal pressures applied.

The principal stress colored contour plot shows that faint brown color indicates the region of moderately stresses (tensile stress) and blue color indicates the region of low stresses (compressive stresses).While red color indicates high stresses. If the red color pattern as seen in the point form is called as singular stresses. These singular stresses are generated due to loading, boundary condition and geometry. The post processing interpretation is done by simply ignoring these singular stresses and neglecting unrealistic stresses.The maximum principal stress 165.55 N/mm2 and 248.325 N/mm2 founds in the rib at 10.34 MPa and 15.51 MPa internal pressures respectively. While

minimum principal stress (2nd principal stress) 24.108 N/mm² and 36.162 N/mm² found at flange corner for 10.34 MPa and 15.51 MPa internal pressures respectively. As the internal pressure acts on the internal effective pressurizing area of valve body, results to expand the valve body .Ribs tries to hold the valve body in original position so ribs subjects to heavy tensile stress. As the internal pressure increases stresses in the valve body increases linearly.

2.2 summarized principal stress data using FEA

The principal stresses (σ_1 and σ_2) at the same points of interest selected for mounting of strain gauge rosette on valve body are found out. The stresses are found at two different cases by applying (10.34 MPa and 15.51 MPa) internal pressures. Following tables shows 1st and 2nd principal stress results found by FEA.

Location Number	1st principal stress N/mm2	2nd principal stress N/mm2
1	36.4	35.6
2	55.4	50.7

Table2 .1 - FEA results at 10.34 MPa internal pressure pr

Location Number	1st principal stress N/mm2	2nd principal stress N/mm2
1	47.8	44.0
2	70.3	67.3

Table 2.2- FEA results at 15.51 MPa internal pressure.

3. EXPERIMENTAL STRESS ANALYSIS

The Experimental stress analysis means determining the stress value on the surface of the valve body when internal pressure is applied. For the same purpose first it is necessary to find out the value of strain on the same surface. There are different methods to determine the values of stress on the valve body surface like strain gauge method, photo elasticity method, moiré fringe and grid method. But it is flexible and more reliable to use the strain gauge technique for experimental stress analysis as the physical model is available.

Table 3.1 - Experimental Results at internal pressure 10.34 MPa applied.

Rosette Number	ε_1 μstrain	ε_2 μstrain	ε_3 μstrain	σ_1 N/mm2	σ_2 N/mm2
1	97	44	95	36.4	35.6
2	127	47	118	48	44

Table 3.2 - Experimental Results at internal pressure 15.51 MPa applied.

Rosette Number	ε_1 μstrain	ε_2 μstrain	ε_3 μstrain	σ_1 N/mm2	σ_2 N/mm2
1	147	65	136	55	51
2	187	68	180	67	70

4. COMPARISON BETWEEN FEA AND EXPERIMENTAL RESULTS

Table 4.1 - Comparison of 1st and 2nd principal stress values at internal applied pressure 10.34 MPa

Table 4.2 - Comparison of 1st and 2nd principal stress values at internal applied pressure 15.51 MPa

Strain Gauge number	FEA results		Experimental results		% Deviation	
	1st principal stress N/mm²		1st principal stress	2nd principal stress	1st principal stress	2nd principal stress
1	86.17	52.4	80.09	49.25	7.05	6.01
2	22.3	14.92	20.20	13.7	9.41	8.17

5. Optimization results of valve body by reducing the wall thickness and flange thickness.

Parameter Changed	Max Vonmises stress(N/mm²)	Reduction in weight (Kg)	weight reduction In (%)
Reducing wall thickness by 1 mm.	235.7	4.125	1.1
Reducing wall thickness by 3 mm.	241.1	5.012	1.4
Reduced wall thickness 1mm and flange thickness by 2 mm	235.5	7.467	2
Reduced wall thickness 1mm and flange thickness by 3mm	219.1	9.138	2.5

Table -5.1 FEA results and reduction in weight after design

Reducing the wall thickness by 1 mm weight is reduced by 4.125 kg (1.1%), while maximum principal stress level value is 235.7 N/mm².Where as when the wall thickness is reduced by 3 mm, weight is reduced by 5.012 kg by keeping the maximum stress value at 241.1 N/mm².Again for another modification, By reducing the wall thickness by 1 mm and flange thickness by 2 mm, the weight reduced is 7.467 kg (02%) from original weight, and maximum stresses found equals to 235.5 N/mm². While if the wall thickness is reduced by 1 mm and flange thickness by 3 mm, 9.38kg (2.5%) weight is reduced and maximum stresses found equals to 219.1 N/mm². Results by decreasing the wall thickness and flange thickness are better than only reducing the wall thickness. Because Principal stresses are not change in flange portion as the thickness of flange is reduces and the principal stress value coming is much lower than the yield stress value, hence designs are safe for working conditions.

6. CONCLUSIONS

1.Results of finite element method for the structural analysis of the gate valve body are well in agreement with experimental results, as the deviation is maximum deviation is up to 2.5 % and minimum deviation is up to 1.1 % which is allowable.

2. Eight new different optimized models created by changing the design parameters and analyzed .As there is restriction to change the flange dimensions and wall body thickness are considered for optimization.

3. Results of decreasing the wall thickness and flange thickness are better than only reducing the wall thickness.

4.The best optimized model is that, in which wall thickness is reduced by 1 mm and flange thickness is by 3 mm reduces 9.138 kg (2.5%) weight, because maximum stress level is much lower than the yield stress value of the material. FEA results for this optimized model shows that stresses in flange are not affect able because of reducing flange thickness.

7. 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 Engg,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 f 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.

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