Thickness Optimization of Valve Plate Used in Flow Control Valve for Pressure Vessels

[1]Mr. A. B. Shaikh, [2]Prof. R. R. Gidde
SVERI's College of Engineering, Pandharpur,India

Abstract: - *Design engineers must use their experience and the latest design tools to maintain reasonable safety levels while providing the most cost effect design. One tool being used on an ever increasing basis is Finite Element (FE) analysis software. A lot of research is being directed towards snap shut and gradually flow reducer valves due to high pressure and high temperature application like fertilizer industries, process industries, boilers etc. these valves act not only as structural element but also source of high stresses in a high pressure application systems. The key challenge to determine high stresses and design parameters is the improvement of dynastic accuracy based upon given amount of information. It is observed that a lot of work has been done on valves. A little work is reported on high stresses in snap shut and gradually flow reducer valves. This work attempts to analyse high stresses and design parameters of these valves and also determine these high stresses by using FEA analysis & finalise the valve thickness. The effect of severity of high stresses on the snap shut and gradually flow reducer valves also need to be investigated. Experiment needs to be conducted, for obtaining high stresses of these valves with different loading conditions and different design parameters. Further, the present research work to enhance snap shut and gradually flow reducer valves reliability, availability, safety and reduce maintenance cost. The scope of dissertation is to analyze design parameters of self-regulating pressure valve and determine which the optimum design solution is.*

Keywords*: Flow Control Valve, Finite Element Analysis Thickness optimization, Leakage Testing.*

1. INTRODUCTION

Most flow applications require regulating the flow of liquid, and usually the parameter of concern is the pressure. This project focuses on the analysis and optimization of gradual flow reducing valve used for safety of a vessel where exchange of liquid between two chambers is taking place where it is required that flow to be shut off when a certain pressure limit is reached. Electronic valves are available, however the intent of this design project is have a total mechanical system, which has an in built response mechanism as shown,

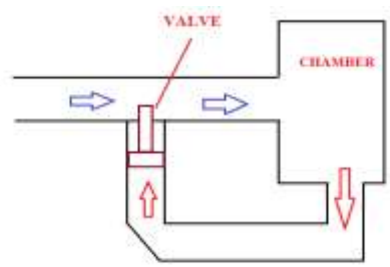

Fig. 1 Valve Position

A lot of research is being directed towards snap shut and gradually flow reducer valves due to high pressure and high temperature application like fertilizer industries, process industries, boilers etc. these valves act not only as structural element but also source of high stresses in a high pressure application systems. However if the gas is toxic, release should be the last option. The idea is then to regulate inflow into the vessel, once a critical pressure is reached. This valve will be an additional fail safe, in case the electronics fail.

1.1 Problem Statement

In gradual flow reducer valve, the obstructer will have a combined load of bending due to the flow, plus the load of the springs and pressure. This will be a transient phenomenon and will need careful FEA for determining life of the valve determining life of the valve. In Snap shut design of the Mechanism will be more Challenging as it should trip at the desired pressure. All valves are not to be used for such high pressure applications due to their low capacity and pressure, temperature limits.

2. DESIGN & ANALYSIS OF FLOW CONTROL VALVE

2.1 Input Data

The available data on field is provided by the client. Depending upon these data design the valve has to be made.

Table 1 Design Inputs of Inlet Loop (Pipe)

Sr. No.	Description	Value	Unit
1	Inlet Radius	20 mm	Mm
2	Inlet Pressure	Min 0.1 and Max 0.32	MPa
3	Inlet error	N.A	-
4	Inlet sensitivity	± 0.01	MPa
5	Inlet fluctuation	Not More than 0.01	MPa /min
6	Operational Temperatures	70 - 350	°C

. **Table 2 Design Inputs of Outlet Loop (Pipe)**

Sr. No.	Description	Value	Unit
1	Feedback inlet Radius	15	mm
2	Feedback Pressure	Min 0.1 MPa	0.32 MPa
3	Feedback error	less than 3%	-
4	Feedback sensitivity	± 0.01	MPa
5	Feedback fluctuation	Not More than 0.05	MPa /min
6	Operational Temperatures	70 - 350	°C

3. DIMENSIONS OF VALVE ELEMENTS

3.1 Analysis of Valve Plate for thickness optimization

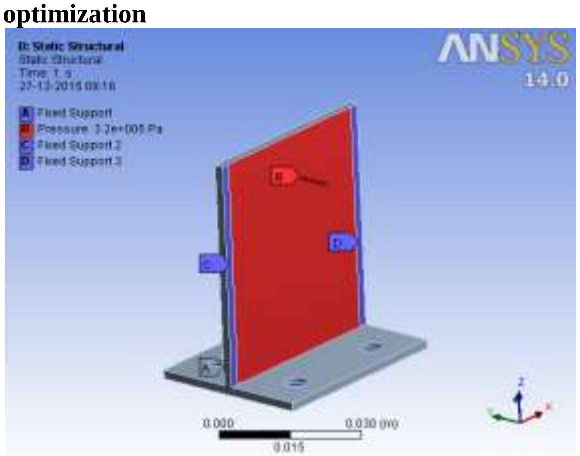

Fig. 2 Boundary conditions

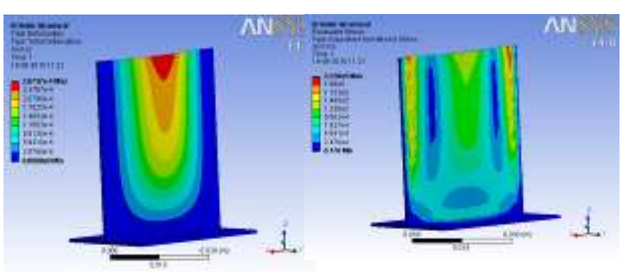

Fig. 3 Total Deformation & Static analysis for Valve plate of 1mm thickness

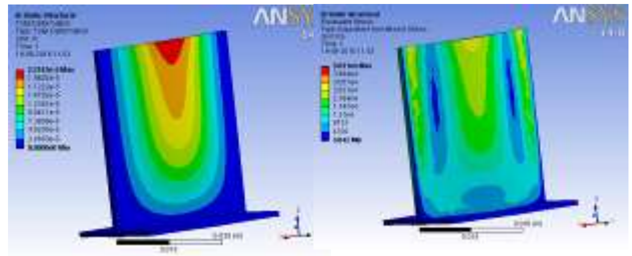

Fig. 4 Total Deformation & Static analysis for Valve plate of 1.5mm thickness

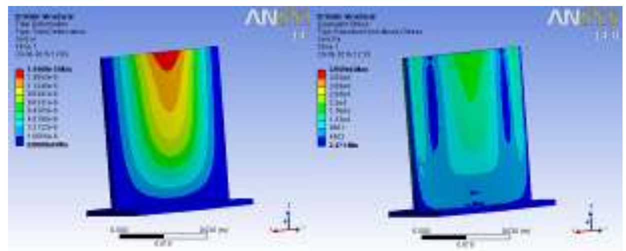

Fig. 5 Total Deformation & Static analysis for Valve plate of 2mm thickness

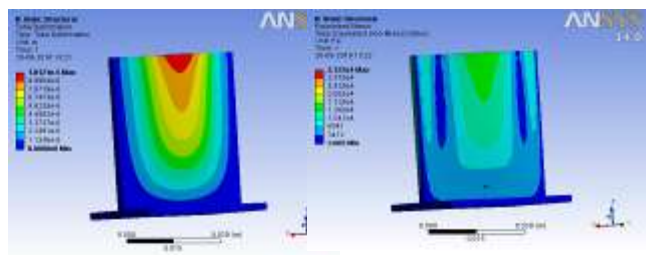

Fig. 6 Total Deformation & Static analysis for Valve plate of 2.5mm thickness

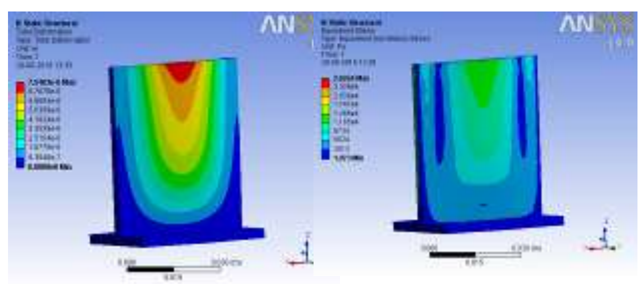

Fig. 7 Total Deformation & Static analysis for Valve plate of 3mm thickness

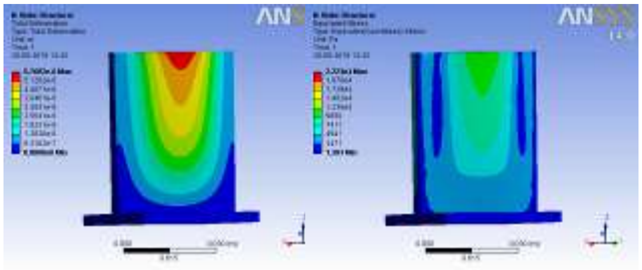

Fig. 8 Total Deformation & Static analysis for Valve plate of 3.5mm thickness

Table 3 Results of thickness optimization analyses when plate is in bending only

Plate	Volume (m^3) 10^-6	Mass (Kg) 10^-2	Total deformation (mm)	Eq. Stress (MPa)
1 mm thick	3.8759	1.7442	0.26737	0.22279
1.5 mm thick	8.1734	3.678	0.02214	0.039307
2 mm thick	10.131	4.5589	0.01446	0.039594
2.5 mm thick	12.088	5.4398	0.01012	0.031223
3 mm thick	14.046	6.3207	0.00754	0.026199
3.5 mm thick	16.003	7.2015	0.00576	0.022229

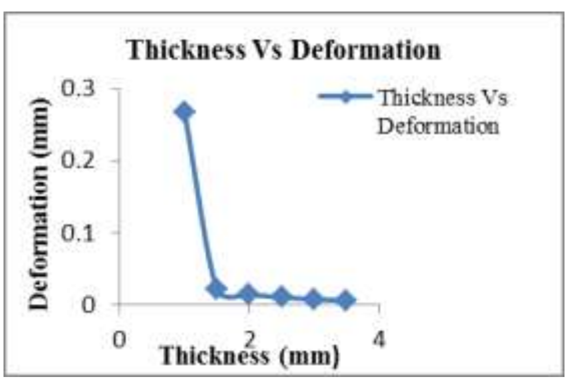

Fig. 9 Graph of Thickness Vs Deformation

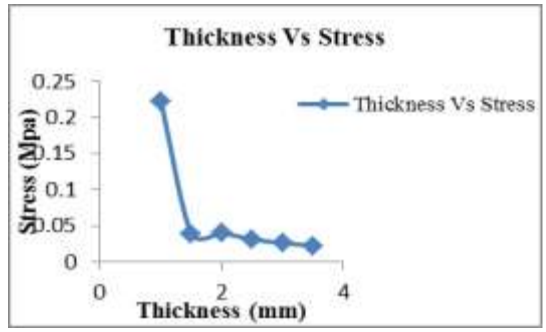

Fig. 9 Graph of Thickness Vs Stress

From above table 3 it is clear that the there is acceptable stress and deformation for 2 mm thick plate. Also the weight and volume is less as compared with 2.5 mm thick plate and 3 mm thick plate. As the stress and deformation values of 2 mm thick plate are very less than that of 1 mm thick plate and 1.5 mm thick plate so it is obligatory to choose the 2 mm thickness for the plate. The graph of thickness Vs deformation from figure 8 is evidence for obtaining minimum deformation

at 2 mm thick plate. The graph of thickness Vs stress from figure 9 is proof for obtaining minimum stress at 2 mm thick plate. The optimization of variable thickness of plate is studied. In particular, 2 mm thick plate all the requirements are gets fulfilled. Hence 2mm thickness selected for the valve.

3.2 Analyses on plate 0f 2mm thickness in various operating conditions

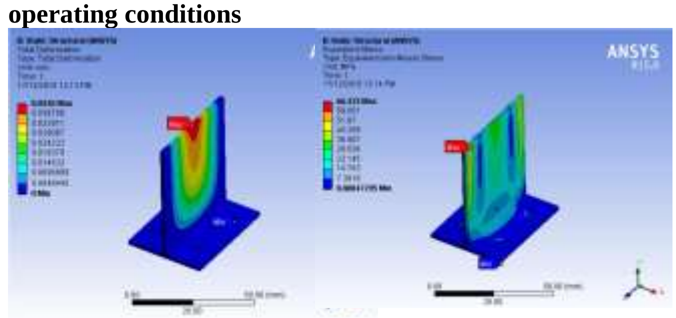

Fig. 10 Total Deformation & Static analysis for Valve plate of 2mm thickness only in Bending

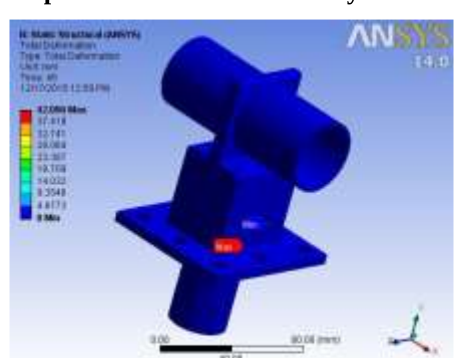

Fig. 11 Deformation of valve for transient analysis

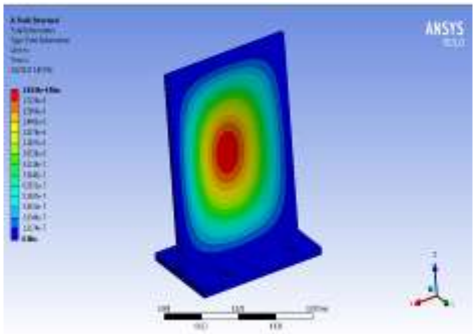

Fig. 12 Deformation in valve plate for 2 mm thickness only in bending with extra supports at top side

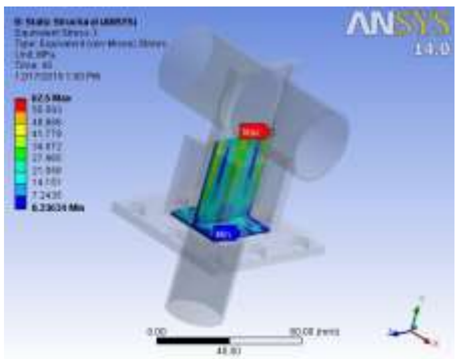

Fig. 13 Stress in only valve plate for transient analysis

Figure 10 shows stress in valve plate for 2 mm thickness in only bending and the value of equivalent stress is 66.433MPa. Deformation of valve for transient analysis is 42.096 mm and it is the travel of plate from bottom to top of jacket, this travel is necessary for full closing of the valve after filling the chamber or pressure vessel. Figure 12 shows deformation in valve plate for 2 mm thickness when titanium plate of valve is subjected to only in bending load and one extra support at the top of plate under 0.32 MPa. The maximum deformation is taking place at the centre of the titanium plate as present in same Figure 12. The value of the maximum deformation in bending load with one extra support at the top is 0.0018444 mm. In Figure 13 equivalent stresses in only titanium valve plate for transient analysis has been shown. The maximum value of the equivalent stress for this analysis is 62.5 MPa.

The second and third modes cause change of shape or size of the component rendering it useless and, therefore, refer to functional or operational failure. Most of the design problems refer to one of these two types of failures. Here for the valve plate if the deformation is not very close to the zero then there is operational or functional failure due chocking of plate in the jacket. Obtained value of deformation by finite element analysis for the titanium plate of valve is 0.0018444 mm which is very close to the zero. Hence analysis done by finite element method is acceptable. These results of FEA can be validated with experimental.

Different graphs from the FEA results are obtained and plotted below.

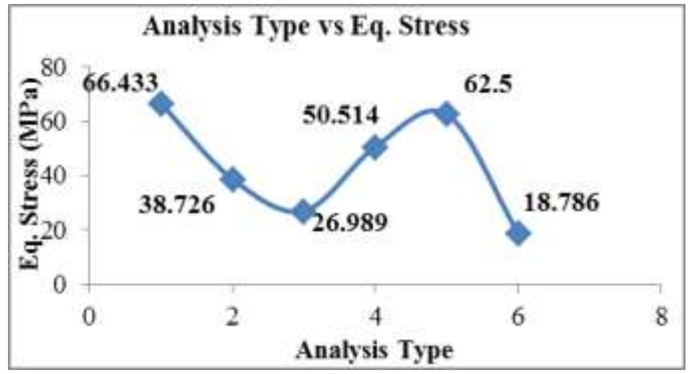

Fig. 14 Graph of Analysis type Vs Equivalent stress

Figure 6.6 is a comparison of different types of analysis with different boundary conditions to which valve assembly is subjected. From the Figure 6.6 and particular result Table 6.1 it is clear that stress is maximum for nonlinear transient analysis, which is equal to 62.5 MPa in vertical pressure & 18.786 MPa in horizontal pressure condition. Also it is clear that stress value is minimum in jacket from jacket stress analysis, which is 26.989 MPa.

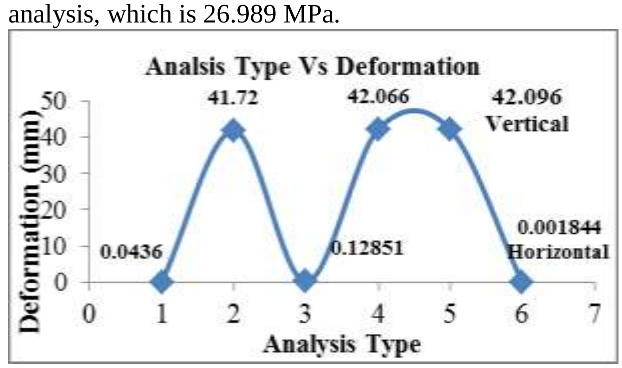

Fig. 15 Graph of Analysis type Vs Maximum deformation

Figure 15 is a comparison of different types of analysis with different boundary conditions to which valve assembly is subjected. From the Figure 15 and particular it is clear that deformation is maximum for nonlinear transient analysis, which is equal to 42.096 mm, but it is the travel of valve plate, hence closing of valve is achieved with this travel against inlet pipe diameter (40 mm). From the analysis of type plate in bending with extra support at top, maximum deformation is 0.0018444 mm which is very closer to zero. These both values (42.096 mm and 0.0018444 mm) are shown in Figure 15.

3.3 Comparison with experimental results

Jacket plate bend test is carried out for finding out the displacement (Deformation) of the plate at center. For this test setup has been made and test is performed on the actual model of the valve. The three trials are taken on the jacket plate bend test and the readings of displacements (deformation) for three trials are as below Table 4

Table 4 Readings of jacket plate bend test

Trial number	Displacement in mm
1	0.00183
2	0.00173
3	0.00187

Average displacement of these three trials is given as,
Average Displacement = (0.00183 + 0.00173 + 0.00187) / 3

= 0.00181 mm

The maximum allowable displacement of valve plate is 0.003 mm, so obtained experimental results of displacement (Avg. Displacement = 0.00181 mm) are within acceptable range.

Operation test is carried out for inspecting the valve for leakage. Here two inspections were conducted, out of these first one is visual inspection and other is ultrasonic inspection. After performing these two tests the following observations were obtained,

1. No visual leaks were detected at peak pressure.
2. Pre and post ultrasonic tests reveal no significant deviation.

4. CONCLUSION

To validate the numerical results obtained from the finite element analysis, the results obtained from experimental jacket plate bend test were collected. The numerical models was developed using the corresponding geometric and material values. Finite element value of maximum deformation is 0.0018444 mm whereas experimental value is 0.00181 mm. The finite element results were increased by near about two percent than that of experimental results. Hence a close agreement between the numerical results and the experimental results were achieved.

REFERENCES

[1] M.R.Mokhtanadeh-Dehghan, N. Ladommatos and T. J. Brennan, "Finite Element Analysis of Flow in a Hydraulic Pressure Valve," Appl. Math. Modelling 1997, 21, pp.437-445.

[2] Binod Kumar, Himadri Chattopadhyay, Pradipta Basu Mandal, Tapas Gangopadhyay, "Dynamic simulation of a pressure regulating and shut-off valve" Computers & Fluids 2014, 101, pp. 233–240.

[3] Qin Yang, Zhiguo Zhang, Mingyue Liu, Jing Hu, "Numerical Simulation of Fluid Flow inside the Valve", Procedia Engineering 2011, 23, pp. 543-550.

[4] Hao Wu, Zheng Zhong, Noureddine Benseddiq, Abdellatif Imad, "Numerical Solution of Crack Shape for the Thermo-Mechanical Loaded Valve," Engineering Failure Analysis 2011, 18, pp. 1487-1495.

[5] Ritika Mohan, Benjamin R. Schudel, Amit V. Desai, Joshua D. Yearsley, Christopher A. Apblett, Paul J.A. Kenis, "Design Considerations for Elastomeric Normally Closed Microfluidic Valves", Sensors and Actuators 2011, B 160, pp. 1216– 1223.

[6] Matin Kagadi., Prof. Girish Tembhare., Vinaay Patil., Sujay Shelke., Optimization of self-activating bi-metallic valve using thermo–structural coupled FEA, Transtech publication inc, publishers in science and engineering, 2012, Pp.147-151.

[7] Greg Ritchie, "Minimizing Pressure Relief Valve Seat Leakage through Optimization of Design Parameters," June 1989, Mass achusetts Institute of technology.

[8] Gabor Licsko., Alan Champneys., Csaba Hos., Nonlinear analysis of a single stage pressure relief valve, IAENG International Journal of Applied Mathematics, 39:4, IJAM_39_4_12, 12 November 2009.

[9] David Kazmer., Mahesh Munavalli., Design and performance analysis of a self-regulating melt pressure valve, University of Massachusetts Lowell.

[10] Gabor Licsko., Alan Champneys., Csaba Hos., Dynamic analysis of a hydraulic pressure relief valve, Proceedings of the World Congress on Engineering 2009 Vol II, London, U.K., WCE, July 2009.

[11] Himadri Chattopadhyay., Arindam Kundu., Binod K. Saha., Tapas Gangopadhyay., Analysis of flow structure inside a spool type pressure regulating valve, Energy Conversion and Management 53, 2012, pp. 196–204.

[12] Professor J. Ed Akin., Design investigation of subsurface safety valves for extreme high pressure and high temperature applications, Rice University, Department of Mechanical Engineering and Materials Science. December 2010.

[13] 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, pp. 469-474.

[14] Deokar Vinayak Hindurao., D.S.Chavan., Optimization of 16" plug valve body using FEA and experimental stress analysis method, International Journal of Mechanical Engineering ISSN: 2277-7059 Volume 1 Issue 1.