International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.3., Issue.6., 2015
(Nov.-Dec.,)

RESEARCH ARTICLE

ISSN: 2321-7758

EVALUATION OF NATURAL FREQUENCY AND MODIFICATION IN VALVE-ACTUATOR ASSEMBLY

ANOOP POKKALATH[1], G.SHAIK USMANSHA [2], DANESH.S[3]
[1]PG Student, Department of Mechanical Engineering, NCERC, Trissur, India
[2]Assistant professor, Department of Automobile Engineering, NCERC, Trissur, India
[3]Assistant professor, Department of Mechatronics Engineering, NCERC, Trissur, India

International Journal of Engineering Research-online (IJOER)
ISSN:2321-7758
www.ijoer.in

ABSTRACT–

In any process plant control valves forms a major element contributing the efficiency and smooth running. This is more critical in power plant safety, especially in region prone to seismic activity. It is normal practice to increase the natural frequency of critical valves above 40Hz to protect the plant from earthquake and similar natural calamities. Lowering valve actuator vibration enhances overall performance, reliability and refinement of an operating plant. Vibration not only has a direct impact on durability and warranty costs but also plays an important. Vibration can also lead to noise, another factor likely to have a significant adverse effect on the perception of overall quality. Increase in vibration can lead to increase the risk of subsequent damage to the components and results in collapse of the system and shutdown. To understand the vibration associated with a valve actuator assembly, it is necessary either to build and test a prototype or to reproduce the system using a model. Ideally, models are statistical equivalents of a material structure and closely represent certainty. Vibration analyses used in models that in roll are based on Computer Aided Design (CAD) model. Numerical study of valve actuator assembly is to be performed to find the natural frequency. Modal parameters are energetic properties of the structure as well as resonant frequencies, the damping and form shapes. Modal relationship helps to fine tune the performance of the actuator assembly. Valve actuator has multi mode of vibration. The main objective of this project is to perform modal analysis of actuator assembly using FEA software to find the applicable mode shape and natural frequencies.

Keywords: Valve, Natural frequency, CATIA, CFD, ANSYS

1. INTRODUCTION

Valve actuator assembly plays a vital role in the successful passage of fluid. But the assembly may show improper functioning in transmission of fluids especially to the areas with earth quakes and other natural calamities. This problem can be solved by increasing the natural frequency of the valve actuator. This project finds the solution for the failure of the valve actuator assembly in the disastrous areas. To succeed this project, an estimation of natural frequency has to be done; thereby we can increase the natural frequency above 40Hz.

The valve considered in this project, is a globe valve used in pipelines to transport water. The globe valve derives its name from the shape of the

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in
Vol.3., Issue.6., 2015 (Nov.-Dec.,)

valve stiff. The rich improvements in the designs and performance of Globe valves are still insufficient to the company in the structure of theory and practice. The phenomenon of flow analysis and Structure of Globe valves is the one in which some more progress can be achieved. One of the major limitations connected with the use of globe valves in liquid purpose is its complex fluid flow. It takes place both in part open and in fully open conditions due to varied reasons.

In areas with natural calamities the valve shows some improper transmission of fluids due to its deformation. So the valve has to withstand higher frequencies. The continuous flow of water causes fatigue failure in the valve, thereby limiting its useful life time. To overcome the problem of subsequent replacement and required maintenance, the sponsoring company seeks to find a suitable alternative material. Therefore, CFD analysis was carried out using the fluid structure interaction technique to simulate various stress and strain diagrams for different valve materials in 3-D. This would help to know which valve material has greater fatigue strength, resulting in greater life time and service of the valve when used in long term services.

2. EXPERIMENTAL PROCEDURE

2.1 Globe Valve

A globe valve is a type of valve used for variable flow in a pipeline, consisting of a changeable disk-type element and a stationary ring bench in a generally in the body. Globe valves are called for their sphere-shaped body shape with the two halves of the body being separated by an internal baffle.

The body is the main pressure containing structure of the valve and the most but easily identified as it forms the weight of the valve. It contains all of the valve's parts that will come in contact with the substance being forced by the valve. The Globe Valve body which is used by the manufacturer is made of Low Carbon Steel which is often called as ASTM A352 LCC in the industries.

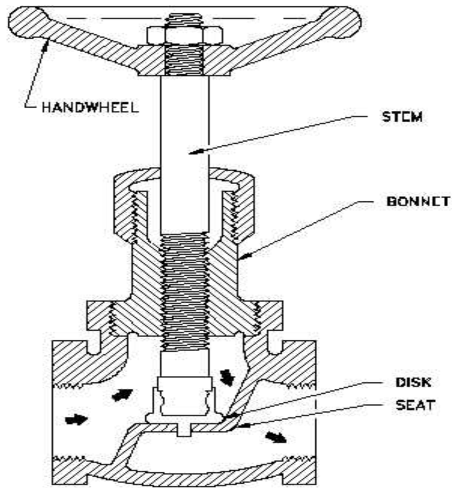

Figure 2.1 Globe Valve Assembly Sketch

2.2 Material

ASTM A352(Low Carbon Steel) This specification covers the low-temperature requirements of service down to 300°F [−184°C] and others down to −425°F [−254°C]. These ranks may be used when crash tested in accordance with Specification A 352/A 352M with energy levels and temperatures of test equally agreed upon between the buyer and the producer.

Table.2 Chemical Composition of ASTM-A352

Ingredients	C	Mn	Si	P	S	Cr	Mo	Ni	Fe
% of Composition	0.06	1.0	1.0	0.04	0.03	11.5 - 14	0.4- 1	3.5- 4.5	78.4- 84.6

Table.3 Material Properties of ASTM-A352

PROPERTIES	VALUE
Tensile Strength(MPa)	750
Yield Strength(Mpa)	495
Youngs Modulus(Gpa)	200

2.3 MODELLING VALVE

2.3.1 SKETCH

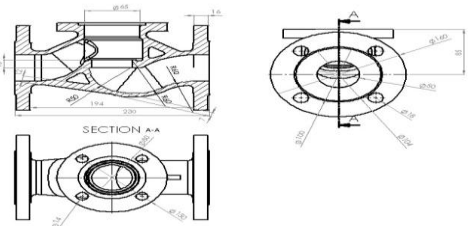

Figure 2.3.1 Drafted Figure of Valve Body

2.3.2 MODAL

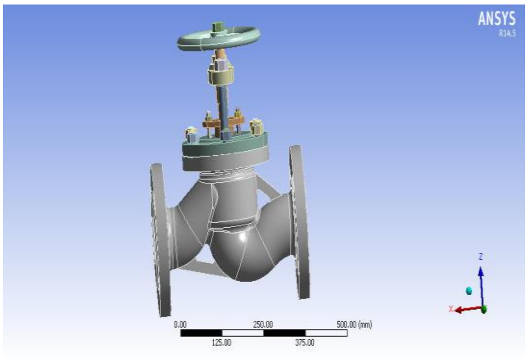

Figure 2.3.2 .CAD modeling of valve

2.3.3 MESHED MODAL

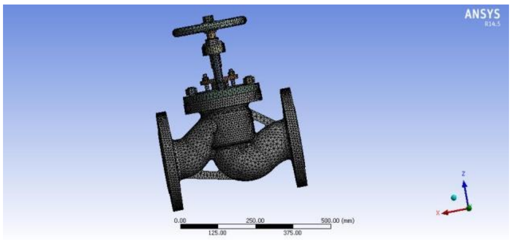

Figure2.3.3 .Meshed model of valve by ANSYS

Table 4 Specification of Valve

Length of the valve	230mm
Inlet Diameter	50mm
Outlet Diameter	50mm
Material	ASTM A352 LCC
Working Fluid	Water
Dynamic Viscosity	1.88 x 10^{-3} Ns/m^3
Density of water	1000 kg/m^3
Inlet Pressure	15 bar
Outlet Pressure	14.656 bar
Inlet velocity	1.87 m/s

3. RESULTS AND DISCUSSION

3.1 CASE 1: CFD Results

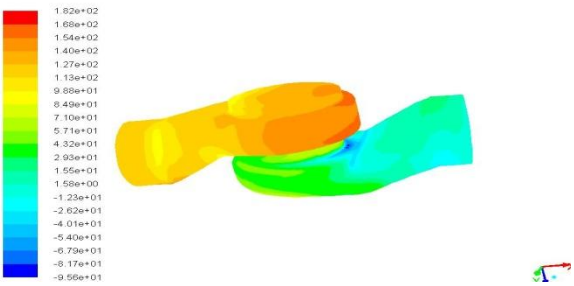

Figure 3.1 Contours of Static Pressure

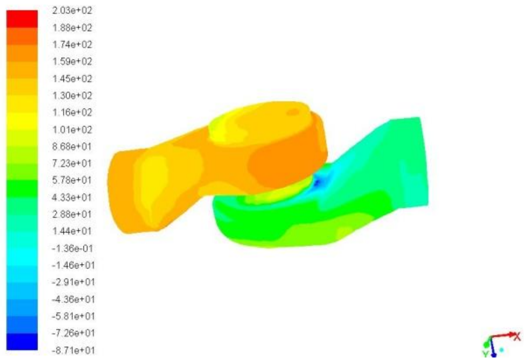

Figure 3.2 Contours of Total Pressure

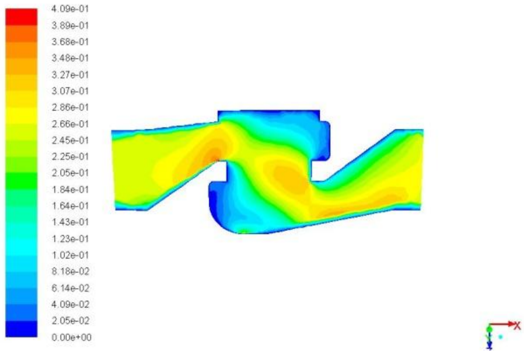

Figure 3.3. Contours of Velocity Magnitude

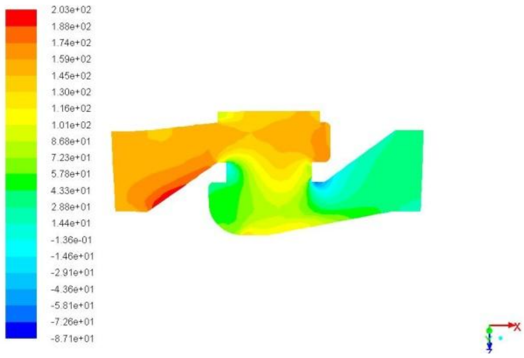

Figure 3.4 . Contours of Total Pressure (Sectional View)

3.2 Full Model Result (Existing With a Thickness of 20 mm)

Here the existing models have a thickness of 20mm. After applying the results from the CFD analysis they obtained pressure is applied here to find the total deformation. The maximum deformation is found at the bottom of the inlet portion of the valve with 0.24169mm.

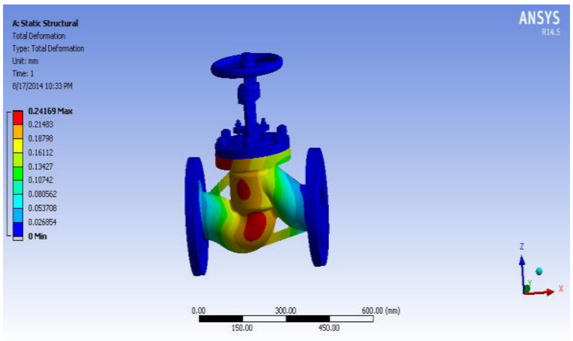

Figure 3.5 . Total Deformation of Existing Model

After deformation then the developed stress are noted. The maximum equivalent stress developed in the model is 368.77 MPa during static structural analysis. The maximum stress developed is at the same region as in the previous case. So the safety factor will be 1.34 which is an admissible one.

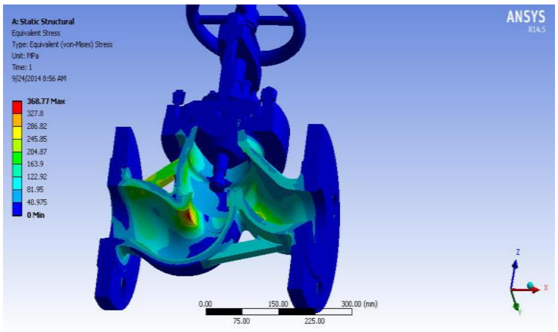

Figure 3.6. Von Mises Stress for Existing Model

With the same conditions the maximum strain developed in the model at the same region is 0.0018439.

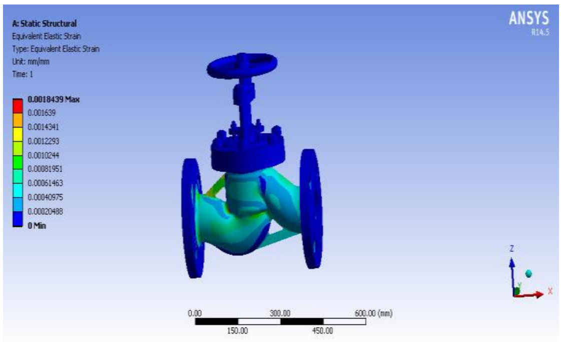

Figure3.7. Equivalent Strain of Existing Model

Under modal analysis the frequency developed for the principal mode is 112.27 Hz and its maximum effect is experienced on the wheel of actuator

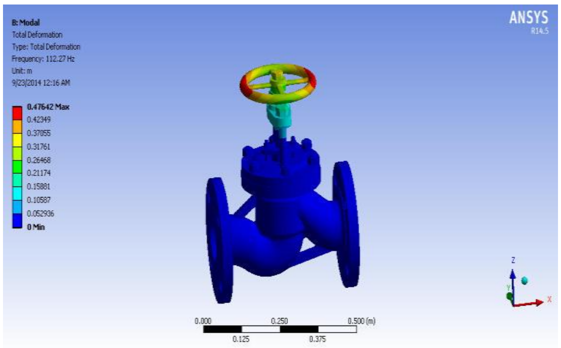

Figure3.8. Principal Mode Shape of Existing Model

The change in maximum stress will be 387.21 MPa in the modified case.

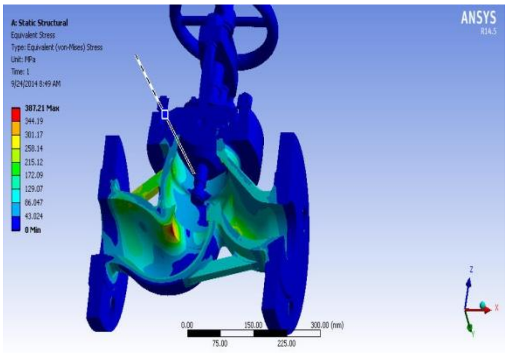

Figure 3.9 Von Mises Stress of Modified Model (10%)

Following stress, the next change observed is in the equivalent elastic strain , which showed a change of 0.0019361 at the maximum.

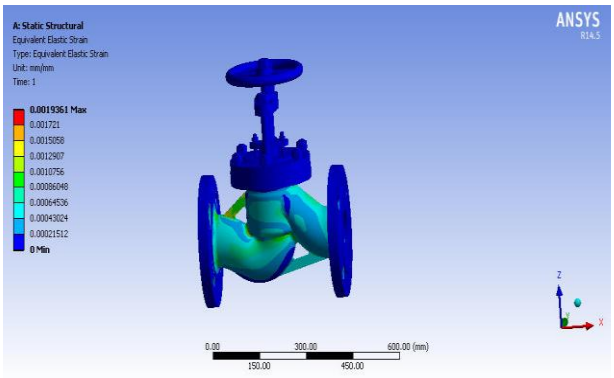

Figure 3.10 Equivalent Strain of Modified Model (10%)

While considering the principal mode shape for the modified valve , the frequency increases to 155.7 Hz. The effect is experienced on the wheel of the actuator which is less while compared to the existing

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in
Vol.3., Issue.6., 2015 (Nov.-Dec.,)

model.

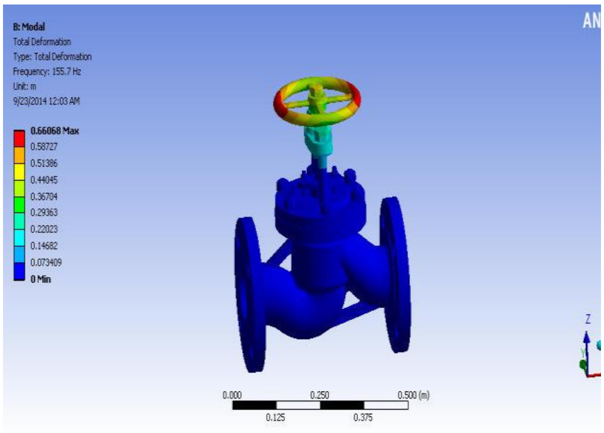

Figure 3.11 Principal Mode Shape of Modified Model(10%)

3.4 Reduction by 20%

After the 10% reduction , the weight again reduced by another 10% . Now the thickness of the model is 16mm.

Now the deformation is changed to 0.26585mm with minor change from the previous modification.

Maximum strain obtained in this case is 0.0020283 which showed a variation from the previous one. The stress is again increased while comparing to the previous model. Now the current stress is 405.65 MPa.

As expected the frequency has been increased after reducing the thickness by 20%. Frequency after modification is 187.57 Hz.

3.4 Reduction by 30%

After reducing the thickness of the model by 30% the new thickness is 14mm.The mere increase in deformation is 0.27794mm. The strain found in this reduction is 0.0021205. Here also the stress increased to 424.09 MPa at the portions as shown in the figure. In the previous modifications the deformation was on the hand wheel portion of the model i.e. at the junction of the valve body and actuator. Now the change has seen on the top coupling portion. The frequency found in this modification is 243.15 Hz.

3.3 Comparison between Existing Model and Modified Models

Table 4 Comparison between Existing Model and Modified Model

Model	Existing	Modified Models		
		10%	20%	30%
Deformation (mm)	0.242	0.254	0.265	0.278
Stress (Mpa)	368.7	387.2	405.6	424
Strain	0.0018	0.00193	0.002	0.0021
Natural Frequency (Hz)	112	155	187	243

4. CONCLUSION

The Project was started with an aim of estimation of natural frequency of valve-actuator assembly. In the beginning a deep study on the valves are made. The valve chosen for the analysis in the company is globe valve. Regarding to it, information related to working, construction and purpose of globe valve are collected. Then the parameters related to experiment specimen are obtained. The available data of parameters related to experiment valve are applied to CFD analysis to identify the contours of pressures and magnitude of velocity. The obtained results are then applied to static structural analysis and modal analysis.

Initially the test has done on valve body which shows a huge change under analysis. After obtaining positive result and identifying the areas of maximum and minimum level, the analysis on full valve proceeded. For having better results the thickness of the body with minimum values are reduced to 10%, thereby reducing the weight of the body. The material of valve is ASTM-A352 LCC. Through this study I came to know that material properties play a significant role in the performance of valve. While considering the material, the valve can show better performance if the grade of the material is ASTM-A316, which offers more corrosion resistance than the existing one.

While concluding this thesis I came to know that the deformations can be reduced by increasing the natural frequency. The reduction in weight is a positive move which is less expensive and showed a better performance than the existing one. In this thesis the modification by reducing the thickness by 10%, 20% and 30% has done. In the future application the idea can be employed for the valve models with higher specifications, especially at the areas prone to frequent natural disaster.

REFERENCES

[1] S.W.Lee,D.Y.Shin,C.W.Byun, H.J. Yang, and I.C. Paek, "Modified Design for the Poppet in Check Valve" Journal of Achievements in Materials and Manufacturing Engineering, Vol.No:23, Issue:1, July 2007, PP:67-70.

[2] K.Soorya Prakash, M.Joseph Malvinraj, and T.Manohar, "Integration of Material Design and Product Design - A CFD Based Approach", International Journal of Dynamics of Fluids ISSN, Vol.No:44, ,December 2009, No:2.

[3] V.J.Sonawane, T.J.Rane, A.D.Monde, R.V. Vajarinkar, ,"Difficulty in manual operation and controlling flow of fluid", IOSR Journal of Mechanical and Civil Engineering , April 2013, PP: 63-71.

[4] Nazih N.Bayomi,Rafea M. Abdel-Maksoud,Mohamed A. Nawar, ,"Valve with Variable Inherent Characteristics" - ISESCO journal of Science and Technology, Vol: 9,May 2013, No:15, PP:39-48.

[5] Lina Bryukhova,"Stress analysis of valves for nuclear power plants''- 20th SVSFEM ANSYS Users' Group Meeting and Conference 2012, PP: 1-7.

[6] Dr.K.H.Jatkar,Mr.SunilShivaji Dhanwe,"Finite Element Analysis of Gate Valve"- International Journal of Engineering and Innovative Technology (IJEIT), Vol:2 , April 2013, PP: 277-281.

[7] Sham Tickoo, "CATIA V5 R17 for engineers and designers''- .Dreamtech press, February 2008.

[8] R. Cohen, Purdue University "Valve Stress Analysis- For Fatigue Problems"-, International Compressor Engineering Conference, No:21,1972, PP:129-135

[9] Schweitzer, Philip A, "Handbook of Valves'' -, Industrial Press Inc., December 1972.

[10] Shapiro, M.B, "Stress Calculation Procedure for Annular Valve Plates in Compressors"- Chemical & Petroleum Engineering, No: 1, January 1965, PP:14-16.