International Journal for Ignited Minds (IJIMIINDS)

Structural Safety and Response Spectrum Analysis of Globe Valve Using Finite Element Analysis

Anand[a] & Yogeesha H C[b]

[a]*PG Scholar, Dept. of Mechanical Engg, Nagarjuna College of Engineering and Technology, Karnataka, India.*
[b]*Asst. Prof, Dept. of Mechanical Engg, Nagarjuna College of Engineering and Technology, Karnataka, India.*

ABSTRACT

Stress analysis plays very important role in the working of the engineering components. Prior stress estimation helps in optimizing the structural design. Also it helps to identify the safety factor in the designs. Due to the advances in the computer based virtual simulation techniques based on numerical methods helps in checking the structural condition of the engineering components under various loading conditions. Also it gives number of alternative designs which helps the designer for better product development.

In the present work, a globe valve of 1.2 meter height is checked for structural safety conditions through finite element analysis. The analysis is extended for spectrum and power spectral response analysis. Initially the geometry of globe valve is built along with its structural components. Later it is imported to Hypermesh for meshing and connectivity. The meshed model is imported to Ansys in 'inp' file format with load collectors. Initially static analysis is carried out based on internal pressure load along with spring tension. The result shows complete safety of the problem. Further analysis carried out to find the rigidity of the valve. The results shows valve is rigid and a static equivalent response spectrum analysis is carried out. The result shows complete safety of the problem along with its structural components. Similarly PSD response analysis results show complete safety of the problem. All the results are represented with necessary graphical plots.

Keywords - *Globe Valve, Structural safety, Response Spectrum, Solid Edge, FEM.*

1. INTRODUCTION

Valves are main elements of mechanical systems where flow control is required. The control may extend to regulate the pressure or mass flow rate or velocity. Valves are mainly used in the piping systems that allow flow of gases, fluids, slurries, vapor etc. Various types of valves exist in the operations such as gate, globe valve, plug, butterfly, check valves etc. Each models has their own significance, advantages and disadvantages. These valves are again has various variety of designs based on the requirements. In this paper, we will discuss about the structural safety and response spectrum analysis of globe valves where the pressure is acting and analyzing seismic condition and structural analysis is carried by using Finite Element Analysis.

2. PROBLEM DEFINITION AND METHODOLOGY

2.1 Problem Definition

Geometrical Modeling and Analysis for structural safety for spectrum loads is the main definition of the problem. The objectives include

- Identification of critical regions of stress concentration
- Improvement by the finite element analysis

2.2 Methodology

- Geometrical Modeling of the valve Structure
- Meshing with appropriate elements (3 Dimensional elements)
- Analysis for Spectrum Loads
- Identification of Factor of safety
- Improvement in the problem and analysis

3. GEOMETRY AND FE MODELLING

3.1 Geometry Modeling

Solid Edge software is used for 3D modeling and dimensioning the globe valve. It has good features to control the precision of dimensioning and along with three principal views an isometric view also can be represented. Geometry is created as per the drawing available, the major dimensions are calculated based in the fluid pressure and working temperature as specified in the Valve standards.

Fig 1: Globe Valve.

The Fig 1, shows Assembly of the globe valve along with the critical parts namely, Valve body, Disk, Bonnet, wheel, stem etc. which directly or indirectly supports the proper functioning of the valve and are assembled to form the globe valve.

3.2 FE Modeling

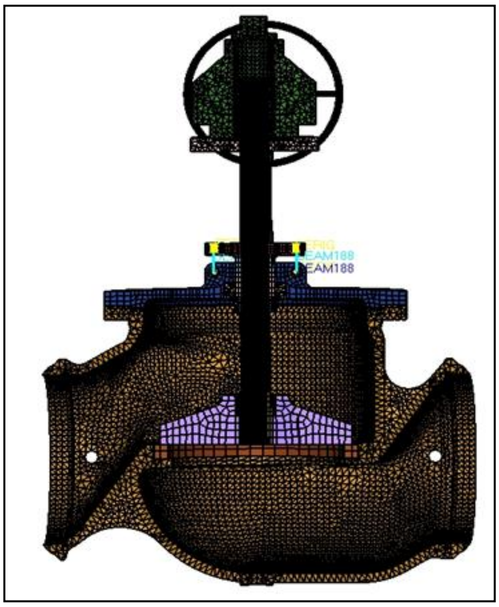


Fig 2: Complete Mesh Plot.

The Fig 2, shows FE model of the globe valve assembly. Simple parts are meshed with 8 noded hexahedral elements and complex geometry are meshed with 10 noded tetrahedral elements, interface connectivity between hexahedral and tetrahedral elements are maintained with the pyramid elements and suitable connectivity between the two parts are assembled by means of bolt connectivity, which is done by using 1D beam elements.

3.3 Boundary Conditions

The Fig 3, shows applied boundary conditions and the loads that are used in the problem. The working fluid is maintained with a fluid pressure of 1.4MPa or 140bar. Hence, this pressure is applied along the fluid path at the inner surface of the valve. Both the ends of the valve are constrained in all the directions shown in the below Fig, since the model is half symmetry, only half model is considered for the analysis and the respective symmetry boundary conditions are applied as highlighted with black color .

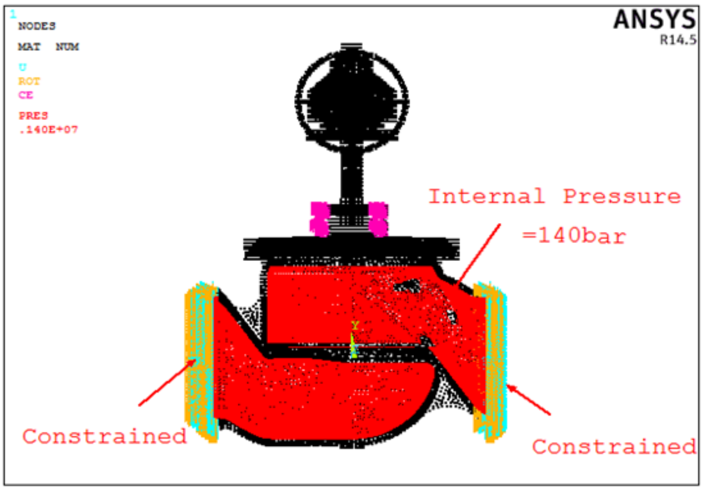
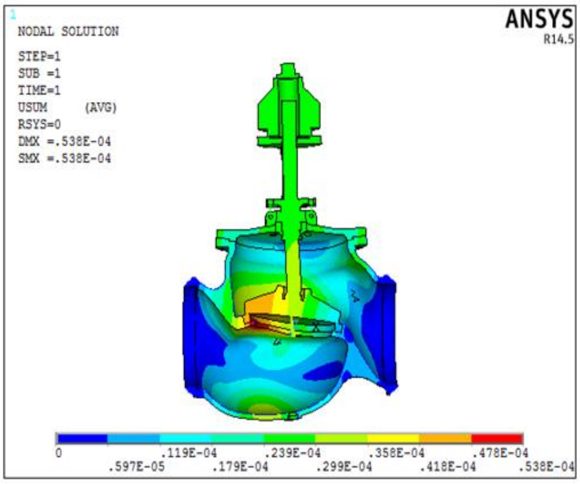


Fig 3: Boundary Conditions of the problem.

4. RESULTS AND DISCUSSIONS

4.1 Static Analysis

The Fig 4, shows the induced deformation in the valve structure for the given static structural loading conditions. From the given Fig 4, the maximum deformation value is observed to be 53.8 microns.



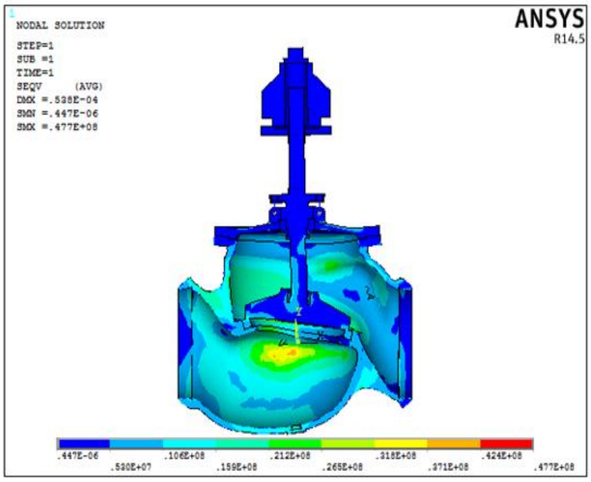


Fig 4: Deformation in the Valve Body. Fig 5: Max von-Mises Stress in the Valve Body.

The Fig 5, shows maximum von-Mises stresses induced in the complete Globe Valve. Maximum von-Mises stresses found to be 47.7MPa at the inner surface of the valve along the flow path, the highlighted stresses are for the given loading conditions. Induced Maximum von-Mises stress region is shown with red color, from contour scale we can observe that apart from the high stress region valve has very less stresses in the complete body. This clearly shows the loads can be further increased. Induced stresses are very much less than the yield stress of the material and the stresses are noticed on a small region, this shows that the current design has the ability to take much higher loads than the prescribed working loads.

4.2 Response Spectrum Analysis

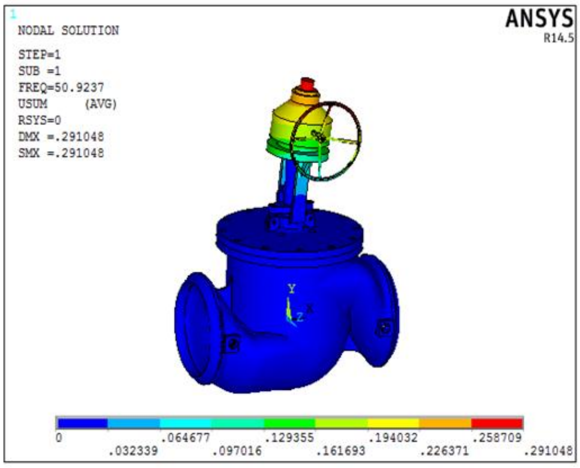


Fig 6: Mode Shape 1.

The analysis results shows first natural frequency value equal to 50.924Hz. This value is compared to find the rigidity of the system as shown in Fig 6.

4.3 Equivalent Response Spectrum Analysis

Maximum acceleration, a:

a =0.16 x 9.81 =1.5696 m/sec^2

Here 0.16 is seismic factor corresponding to Zone3

Internal pressure load, P =140 bar

Gear mass, m = 45 kg

Load factor to be considered =1.6 in all directions.

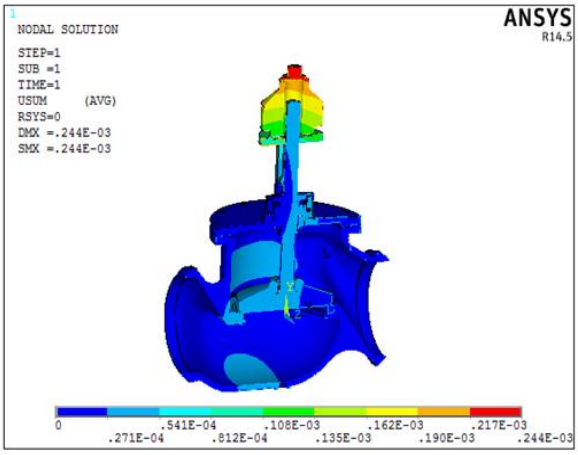


Fig 7: Deformation plot for X-Axis Acceleration.

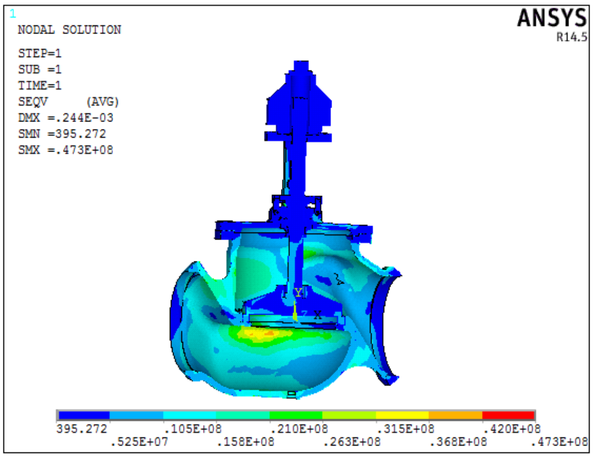


Fig 8: von-Mises Stress plot for X-axis response.

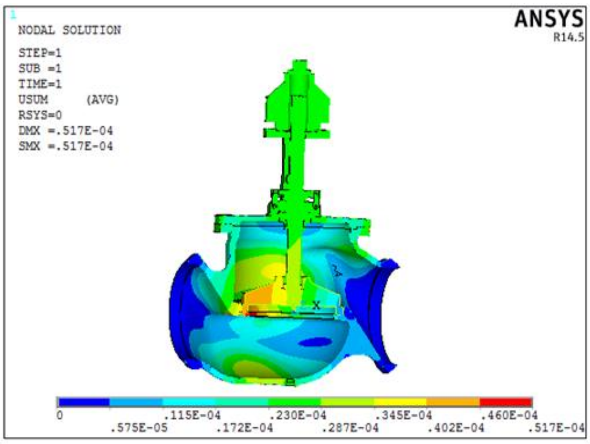


Fig 9: Deformation Plot for Y-Axis Response.

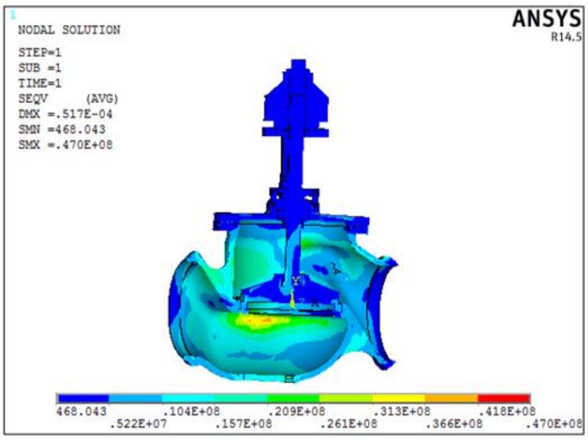


Fig 10: von-Mises Stress for Y-Axis Response.

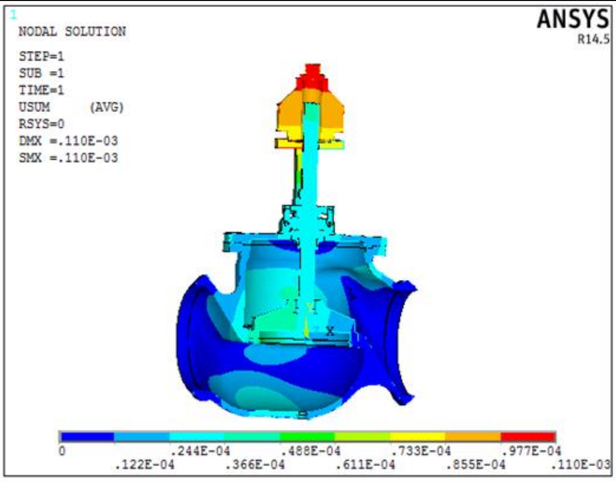


Fig 11: Displacement Response for Z axis Response.

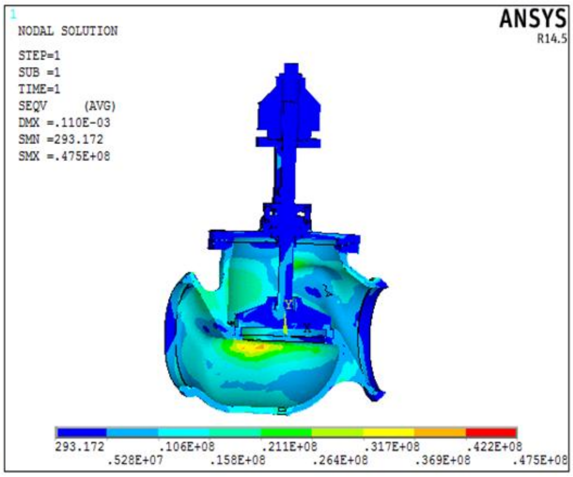


Fig 12: von-Mises Stress Plot for Z axis response.

The table 1, shows maximum stresses are happening for Z-axis acceleration. Also the stress values are almost near to all types of acceleration indicating the uniform strength of the structural members of the valve assembly as shown in Fig 7 to Fig 12.

Acceleration Direction	**Deformation, mm**	**von-Mises Stress, MPa**
X-Axis Acceleration	0.244	47.3
Y-Axis Acceleration	0.05	47
Z-Axis Acceleration	0.11	47.5

Table 1: Displacement and von-Mises Stress for Seismic Equivalent Analysis.

4.4 PSD Response Analysis (Power Spectral Analysis)

PSD analysis is another important analysis to find the safety of the structural components. Modal Analysis Results: Modal frequencies are obtained for the operational range of 1200Hz. Totally 22 sets are required to obtain to match the operational frequency of 1200Hz.

Set No	**Frequency(Hz)**
1	50.924
2	70.187
3	139.55
4	160.36
5	189.36
6	194.86
7	300.21
8	308.29
9	474.76
10	512.2
11	517
12	573
13	610
14	623.77
15	644/0
16	706.61
17	737.84
18	778.92
19	816.34
20	1088.4
21	1123.2
22	1320.6

Table 2: Modal Frequencies for the Configuration.

The table 2, shows increasing order of the natural frequencies. Generally the dependency of maximum value of the natural frequency depends on the input data for spectrum analysis.

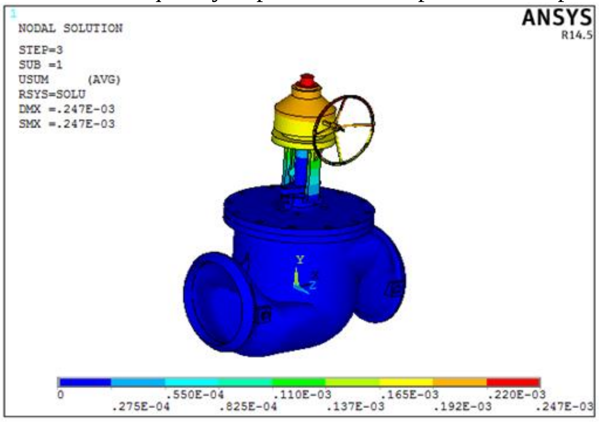


Fig 13: Displacement Plot.

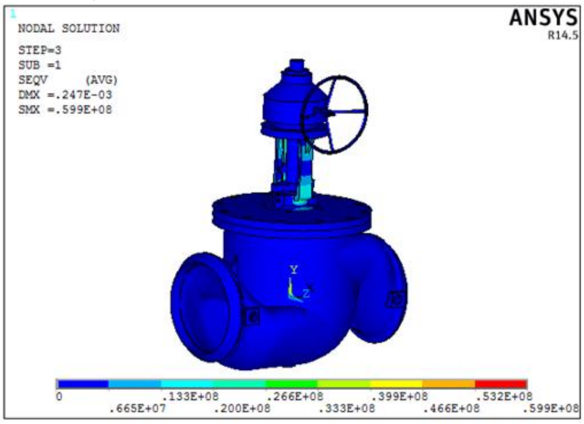


Fig 14: von-Mises Stress Plot.

5. CONCLUSION

- Initially globe valve assembly is built for the specified dimensions using three dimensional CAD software. Initially the individual members are created and assembled using assembly option of the software.
- Initially the meshed geometry is applied with static boundary conditions and structural safety is checked through deformation and stress. 14 bar pressure is applied for the inner body of the structure along with spring tension load of 1200N.
- The results show the stress development of 59.9Mpa for one sigma value. When calculated for 3Sigma values the stress value is still less than the yield stress of the valve components. So the globe valve is safe for 99.7% as specified through power spectral density analysis. All the results are represented through necessary graphical plots.

REFERENCES

1. John Peters, "*Assessment of valve failures in the offshore oil & gas sector*", Research Report 162, Health and Safety Executive, 2003
2. D. Lal, "*The use of Finite element Method for Stress analysis of Compressor Valves*", International Compressor Engineering Conference, 1978
3. R.W. Park, "*Contamination Control – A Hydraulic OEM Perspective*", Work shop on total contamination control, Center for Machine condition Monitoring, Monash University, August 1997
4. M. Hari Prasad, "*Reliability prediction of control valves through mechanistic models*", Journal of Mechanical Engineering Research Vol. 2(3), pp. 52-57, August 2010
5. Dr. K.H. Jatkar, "*Finite element analysis of Gate valve*", International Journal of Engineering and Innovative Technology (IJEIT) Volume 2, Issue 10, April 2013
6. P. Ebenezer, Sathish Paul, "*Design and Analysis of Industrial Ball Valve using Computational Fluid Dynamics*", IJRTME- ISSN: 2347-7326, 2015
7. Karan Soni, "*Optimising IC Engine Exhaust Valve Design using Finite Element analysis*", IJMER, ISSN: 2249–6645, Vol. 5 Iss. 5 May 2015.
8. Leila Ramezani, "*The Challenge of Air Valves: A Selective Critical Literature Review*", American Society of Civil Engineers, 2015
9. Sallet, "*Flow inside a typical safety valve was studied by visualization of the flow in a 2D valve model*". 1981
10. Singh, Dynamics and stability of spring-loaded safety relief valves. 1982.