Numerical Investigation of Manifold under Seismic Conditions

Vidyaprasad Desai
Student, Mechanical Department
Sinhgad College of Engineering
Pune-411041

Dr. Deepak Watvisave
Associate Professor, Mechanical Department
Sinhgad College of Engineering
Pune-411041

***Abstract*—Seismic analysis is the field related to the geological surveying methods involving vibration analysis from natural and artificial circumstances. Seismic analysis is essential for the components to be used in the regions prone to earthquake. It is necessary to improve the performance of the valve parts under varying seismic loading. The project work is based on the seismic analysis of instrumentation valves and manifolds. The valves to be modeled and analyzed will have high working pressures, mostly nuclear conditions, high temperatures and varying sizes. The manifolds to be analyzed will be 3 and 5 valve and various other configurations having high working pressure. The experimental validation for the project will include testing of needle and ball valves. The numerical simulation using FEM will include Static stress analysis and Seismic analysis. Modeling of the valves and manifolds will be done using CAD software and simulation will be carried out using FEA software. The results from static stress analysis and seismic analysis will indicate the allowable working strength for manufacturing the valves and manifolds.**

Keywords—Seismic, Manifold, Nuclear Safety, Modal analysis, Response spectrum

I. INTRODUCTION

The availability factor of nuclear power plants has been significantly improved worldwide, where by nuclear power is becoming more economically competitive with fossil fuels for base-load electricity generation in many countries. Nuclear power accounted for nearly 35% of domestic production electricity in 2012 and is gradually increasing in dependency. Structural integrity includes valves that can withstand earthquakes and other dangerous situations, because they are used to control cooling waters in the primary coolant loop to prevent boiling within the reactor in a nuclear power plant. An accident due to the leakage of radioactive matters, however, can inflict catastrophic damage on the environment nearby. Therefore, with the enhanced awareness of the potential of an earthquake to cause such damage, qualifying the valves has now become standard practice, that is, establishing their ability to withstand a seismic load without damage. Strict safety guidelines should be carried out as defined by the KEPIC MFA, which indicates the verification of seismic adequacy with prescribed safety rates for structures and equipment. The verification of the seismic estimation in order to overcome the limitations when using a finite element model for seismic analysis .Seismic analysis is the field related to the geological surveying methods involving vibration analysis from natural and artificial circumstances. Seismic analysis is essential for the components to be used in the regions prone to earthquake.

The Seismic qualification deals with checking mechanical integrity of the product under the unforeseen event of earth quake. The special emphasis is given to interface of the product with rest of the environment i.e. valve assembled and fixed with the pipes through threads or welding. The stresses developed in the threads and welding under seismic loading need to be checked. Other important aspect is the rotating parts of the valve interacting with stationary parts through tight clearances in parts. The rotating assembly and stationary assembly may have different modes of vibration and they may be out of phase also. It is essential to check relative deflections against minimum available clearances.

II. PROBLEM STATEMENT

The manifold is used to equalize the pressure of fluids. Working pressure of the manifold is in the range of 5 bar to 9 bar and temperature is in the range of -180C to 2600C. Seismic conditions are unpredictable in sense of its acceleration and the frequency. If the manifold is subjected to such unpredictable conditions it can lead to the leakage which can cause the damage not only to power plant but also to nearer human society. So it is very important to ensure the safety of manifold under seismic condition.

III. OBJECTIVE

- The objective of the seismic analysis is to analyze the structural integrity of the valve.
- The mesh quality should comply with the standards and node connectivity has to be established to ensure the close volume geometry.
- The static structural analysis is expected to determine the behavior of the valve under initial boundary conditions of loads and pressure.
- The modal analysis is required for 18 modes to calculate the corresponding mode shapes of the valve
- The response spectrum analysis is required to calculate the accelerations of the valve in all three directions for OBE and SSE conditions of loading.
- The manifold has to be tested for various material combinations to check its mechanical integrity.
- The materials to be used are according to various established material standards.

- The valve model will be tested under the required conditions
- The results from the experimental data and analytical data will be compared to get accurate results.

IV. MANIFOLD DESCRIPTION

The Fig 3.3 shows 3-valve configuration which is available for use with differential pressure and multi-variable transmitters. Two block valves provide instrument isolation, and one equalize valve is positioned between the high and low transmitter process connections.

V. NUMERICAL ANALYSIS

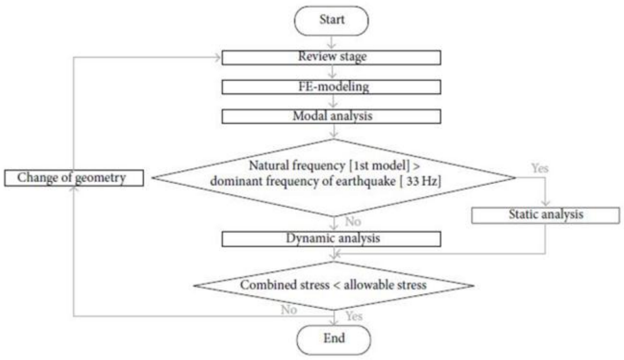

Fig. 1. Flow of Numerical Analysis

The history of the earth quacks at the nuclear power plant was studied and it was observed that the maximum frequencies of earth quacks until now is up to 33 Hz only. It implies that critical frequency of earthquake in such zones is 33Hz. So it was generalized that if the first mode of component is less than the 33 Hz it needs to be analyzed dynamically for the response spectra otherwise its safety can be justified by its static analysis only. So it becomes pre-requisite to do modal & static analysis. If the first mode comes below 33Hz then we have to go for dynamic analysis.

A. Geometrical modelling

The CAD geometry of Straight-type manifold was modelled in SolidWorks 2013. The geometry was divided into two parts, sub-assemblies 1 and 2. Sub-assembly 1 contains the Body which includes passages for fluid flow. Sub-assembly-2 includes handle, spindle, check nut, bonnet, gland, gland seal, gland nut, washer, grub screw and ball.

The geometry file created in CAD software is saved in STEP format. The STEP format is preferred over IGES as it results in less data transfer loss. STEP file is supported by FEA software. In this dissertation, the sub-assemblies 1 and 2 are saved separately for meshing ease.

B. Meshing And Convergence

The geometry is meshed in the Hypermesh with brick meshing. The convergence is checked for the optimize size of the elements. The Convergence is achieved by h-refinement. The minimum variation in the stress is observed at the element size of 2 mm. It concludes the convergence is achieved at 2 mm.

VI. NUMERICAL INVESTIGATION UNDER STATIC CONDITION

The static structural analysis is carried out to verify the manifold's safety at the static condition of operating pressure. The operating pressure of manifold is 9.44 bar (0.944 MPa).The static structural uses simple F=K*x formula to calculate the displacement of an individual node. From the displacement matrix the strain matrix and further stress matrix is calculated.

The post processed result contains the equivalent stress, strain and the total deformation. Equivalent stress is von mises stress. Von mises stress are calculated by using an Energy distortion method.

According to the von Mises's theory, a ductile solid will yield when the distortion energy density reaches a critical value for that material. Since this should be true for uniaxial stress state also, the critical value of the distortional energy can be estimated from the uniaxial test. At the instance of yielding in a uniaxial tensile test, the state of stress in terms of principal stress is given by: $\sigma 1 = \sigma^* Y$ (yield stress).

A. Result and Discussion-

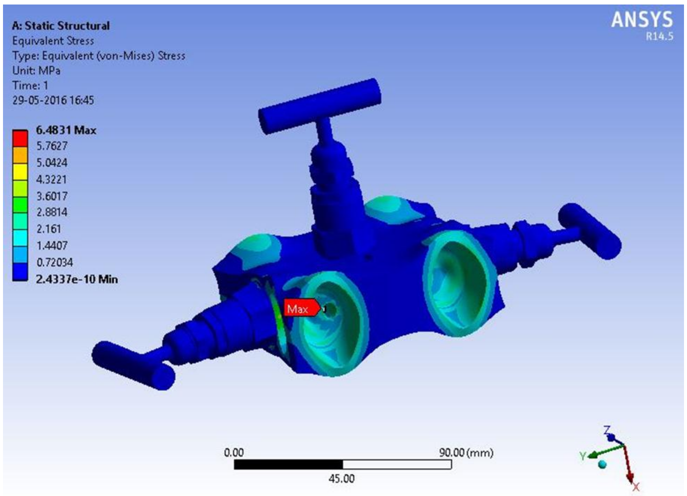

Fig. 2. Von-Mises Stress in Manifold

The maximum equivalent stress is at the edge where the fluids are mixed the stress will increase further if the operating way gets block. From the situation we can conclude that the maximum stress is well under the limit. The factor of safety provide is much greater. There may be variation in the pressure of operating. The maximum variation will be in range of 15% of operating one. The higher value of factor of safety will accommodate these variations. At the inlet of a manifold the cross section changes suddenly. The comparatively higher stress at inlet can be justified by this.

B. Summary-

- The stresses are within the limits. The maximum stress is much less than the allowable stresses. So the manifold is over safe, and can be optimized.
- The maximum stress occurring is nearer to the holes which are constrained. So we can say that the location of holes is selected effectively

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 09, September-2016

VII. MODAL INVESTIGATION

Constraints- Except free run, natural frequency being inherent characteristic property of any component /assembly no external force is applied during the analysis

Damping is neglected in modal analysis

Output from modal analysis- frequency of vibration and corresponding displacement

If the first several frequency comes 0 it shows the improper connection between the parts.

The modes corresponding to 0 frequency are called as rigid modes and they are neglected.

A. Result and Discussion-

B.

TABLE I. MODES OF MANIFOLD AND RESPECTIVE FREQUENCIES

Mode	Frequencies by FEA	Description of Mode
1	29.407	First Twisting Mode (about X-axis)
2	129.74	Second Twisting Mode (about Y-axis)
3	226.36	Third Twisting Mode (about Y-axis)
4	377.22	Fourth Twisting Mode (about X-axis)
5	500.87	Fifth Twisting Mode (about Y-axis)
6	568.81	Sixth Twisting Mode (about Y-axis)
7	576.44	Seventh Bending Mode (about Z-axis)
8	606.96	Eighth Bending Mode (about X-axis)
9	612.72	ninth Bending Mode (about X-axis)

B. Summary-

- The Fundamental eigen value come 29.37 Hz which is less than 33 Hz. So it becomes necessary to do Response spectrum analysis.
- First 6 mode shapes ate twisting mode shapes.
- Last 3 mode shapes are bending mode shapes.
- If the operated at the last three natural frequencies the spindle may fail.
- Only first mode is considered as critical frequency for earthquake till now is 33 Hz only.

VIII. RESPONSE SPECTRUM METHOD

In order to perform the seismic analysis and design of a structure to be built at a particular location, the actual time history record is required. However, it is not possible to have such records at each and every location. Further, the seismic analysis of structures cannot be carried out simply based on the peak value of the ground acceleration as the response of the structure depend upon the frequency content of ground motion and its own dynamic properties. To overcome the above difficulties, earthquake response spectrum is the most popular tool in the seismic analysis of structures. There are computational advantages in using the response spectrum method of seismic analysis for prediction of displacements and member forces in structural systems. The method involves the calculation of only the maximum values of the displacements and member forces in each mode of vibration using smooth design spectra that are the average of several earthquake motions.

A. Result and Discussion-

The manifold is tested for OBE and SSE along each axis. The Critical results were OBE-Z, SSE-X

- **OBE-Z** INVESTIGATION-

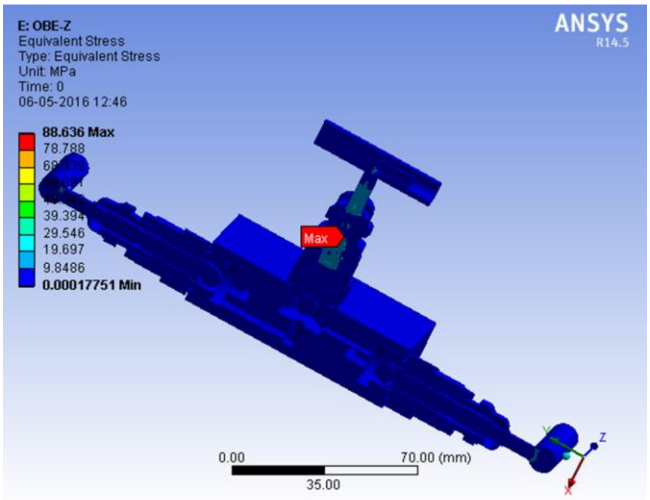

Fig. 3. Von-mises stress in manifold at OBE-Z Condition

- The spindle aligned in Z –direction has higher stresses.
- The allowable stress for spindle is 168 MPa.
- The spindle's cross sectional area where the sudden change is there has higher stress.
- The spindle is safe for OBE-Z seismic condition.
- **SSE-X** INVESTIGATION-

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 09, September-2016

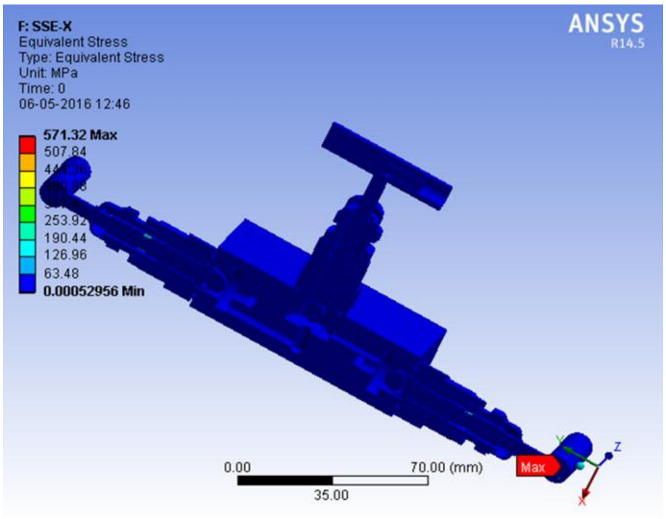

Fig. 4. Von-mises stress in manifold at SSE-X

- This result follows very different pattern than the other patterns.
- The max stress is 571.32 MPa which exceeds the allowable stress
- The location of this stress is at the contact area of lateral left spindle and handle. So the handle will fail as it is weak component.
- Failure of handle won't cause any leakage. But the spindle wouldn't be tuned off.

IX. CONCLUSION

- Manifold is safe under static condition.
- First six modes are twisting modes and further 3 modes are bending one.
- Manifold can fail for SSE-X at contact area of spindle and handle and safe for other all excitations by FEM.

X. FUTURE SCOPE

- The Manifold is over safe in static condition, where as it is failing at SSE-X conditions.
- The material should be effectively distributed to avoid the failing of manifold under SSE-X.
- The topological optimization of main block of manifold could be carried out as it is over safe.
- The material optimization is optional way to save the cost.
- The manifold is operated in cryogenic conditions so it becomes necessary to analyze the manifold under those conditions.
- If manifold is operated under its natural frequencies the spindle will fail by bending. The spindle should be redesigned to prevent the failure.
- The manifold block can be meshed with hexa element to increase the accuracy and to decrease the computational time.

REFERENCES

[1] Sang-uk Han,Structural 'Safety analysis on seismic service condition" Dong-A University, Busan , Republic of Korea, Published 11May 2014.

[2] Dai-Ye, "Response spectrum derivation from floor"Harbin University of Science and Technology, Heilongjiang Harbin 150080, China

[3] K.K.F.Wong,"Seismic spectra anlysis" Procedia Engineering 14(2011)1645-1652, National Institute of Standards and Technology,USA

[4] Claudio Braccesia, Procedia Engineering 101 (2015) 493 – 500, 3rd International Conference on Material and Component Performance under Variable Amplitude Loading.

[5] Ingrid Delyová, Peter Sivák, MMaMS 2012, Procedia Engineering 48 (2012) 77 – 82.

[6] M. Stosiak ,Wrocław University of Technology, Faculty of Mechanical Engineering, 50-371 Wrocław Stosiak.

[7] Tom Irvine* NASA Engineering & Safety Center (NESC) Loads & Dynamics Technical Discipline Team.

[8] M. Fossati,Modal analysis of a component, Milano, Italy

[9] C. Azoury, Experimental and Analytical Modal Analysis of a Crankshaft, IOSR Journal of Engineering Apr. 2012, Vol. 2(4) pp: 674-684

[10] Steven E. Benzley, A Comparison of All Hexagonal and All Tetrahedral Finite Element Meshes for Elastic and Elasto-plastic Analysis, Brigham Young University Provo, Utah.

[11] Dr. Suresh P M, "Experimental Modal Analysis of Automotive Exhaust Muffler Using Fem and FFT Analyzer", .ijrdet Volume 3, Issue 1, July 2014.

[12] Oludele Adeyefa, Finite Element Analysis of Von-Mises Stress Distribution in a Spherical Shell of Liquified Natural Gas (LNG) Pressure Vessels, scientific research 2011, 3, 1012-1017.

[13] B.V. Subrahmanyam, Static and Dynamic Analysis of Machine Tool Structures, IJRMET Vol. 4, Issue spl - 1, No V 2013- April 2014.