Multiphysics Stress and Vibration Analysis of Pressure Vessels under Low Pressure and Medium Temperature Conditions

Shailesh Gawand[1], Santosh R Shekokar[2], Nitin A Kharche[3], Ramakant M Choudhari[4], Pravin P Pande[5], Jivan Ingle[6]

[1]*P.G. student, Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India*

[2,3,4,5,6]*Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India*

Abstract

The work focus on FE analysis and investigations of thermo mechanical constraints on the pressure vessel. The thermal stresses, deformation developed in the finite element analysis (FEA) for the pressure vessel are compared with analytical results. The vessel is designed using ASME section VIII Division 2 and the required thickness of the head, cylindrical shell etc. are to be validated using the FEA tool. The static and thermal analysis (multi-physics analysis, combined loading) will be carried out using ANSYS tool. Also, natural frequency of pressure vessel will found out using modal analysis. Study focuses on two different constraints conditions of the right-hand saddle and performed for constant internal design temperature and internal design pressure. It is found that the pressure vessel is in a safe zone when the right hand saddle is free along axial direction.

Keywords: Failure, Oil, and gas industry, FEA

Introduction

A pressure container is a device designed as a closed container that keeps gases or liquids under pressure that differs considerably from ambient pressure. Due to the different operating conditions of vessels under pressure, they are potentially dangerous and accidents in which they can be deadly and pose fatal risks. The main purpose of this work is to design and analyse a pressurized tank to work under different operating conditions and to identify the most contributory parameter that regulates the efficient operation of the oil tank. In general, the pressure will develop in the oil tank, and will also withstand more forces deployed because of the internal and external pressure working on it, making the design critical. As a result, the pressure vessel is designed for safety reasons in accordance with ASME standards. Additional validation of the analysis software was created by comparing the results with the results of the manually calculated design values. We are working to understand the different loads that have developed in the ship under pressure. Creating a 3D model and analyzing using an appropriate solution. The result was compared with the results of the analysis software and turned out to be with a pleasant range. A Pressure Vessel is a container which contains either external or internal pressure, by the heat applied from direct, indirect or combination of sources. Design of pressure vessel is based on standard technical specification [ASME Code, Section VIII].

Pressure vessels are used for various purposes such as nuclear reactor vessels, pneumatic reservoirs and storage vessels of liquefied gases (Lee et al., 2017). From last few decades due to increased demand of alternative fuels generates the need of high pressure and temperature vessels for petroleum refineries and chemical plants. Pressure vessels are containers used to handle fluids which are highly toxic, compressible and works at high pressures. These vessels are applied in numerous industries such as oil, gas, petroleum, beverage, chemical, power generation, food and fertilizer, etc. Currently there is much advancement in the pressure vessel field like in case of investigating new grade material, composite materials, welding techniques, etc. The applications of finite element analysis is important for understanding of fatigue and creep process (Raffiee et al., 2018). In some chemical industry mixing of liquid or gaseous chemical take place in enclosed chamber such as pressure vessel hence it is called as mixing chamber. For deeper understanding of stress can be archived by multi-physics analysis. The multi-physics analysis is used to determine structure under the action of any combine loads. It is used to determine displacements, strains and force of a component. Hence in present investigations an attempt is made to carry out stress analysis to determine the stresses in structure under the action of static and thermal loads.

In pressure vessel during the operation chemicals comes through the nozzle with different pressure and temperature which varies with respect to time. The change in pressure and temperature results deformation and distortion with high local stresses in mixing chamber which in turns reduces the fatigue life of mixing chamber. The transient dynamic analysis is used to find stress and deformation in the pressure vessels. Hence in present investigations an attempt is made to carry out stress analysis to determine the stresses in structure under the action of any general loads. Also the fatigue life is predicted and enhanced by finite element analysis to determine the time varying displacements, strains and force acting on pressure vessels. The various types of pressure vessel are discussed in next section; the Figure 1 shows the generalized diagram of pressure vessel.

Literature Review

Apurva R.Pendhaje et al (2014) reported that following these standards leaves the designer free from designing the components. This aspect of design greatly reduces development time for a new pressure vessel.

Adithya.M and Patnaik.M. (2013) carried out FEM analysis and reported that design by analysis is most desirable to evaluate and predict the behavior of different configuration of PV supported on saddle with/without stiffener rings.

Vishal V. Saidpatil and Arun S. Thakare (2014) studied design and analysis of PV used in boiler for optimum thickness, temperature distribution and dynamic behavior using FEM. Farhad Nabhani et al (2012) conducted the work on Reduction of stress in cylindrical A pressure container is a device designed as a closed container that keeps gases or liquids under pressure that differs considerably from ambient pressure.

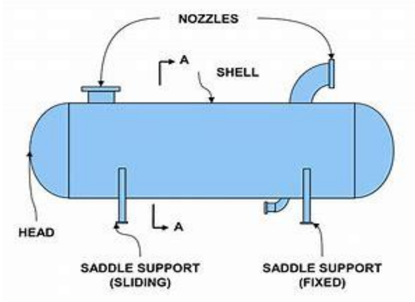

Figure 1. Different components of pressure vessel

Sanal (2000), conducted nonlinear finite element analysis of pressure vessel. In this paper focused on two pressure vessel problem where the large displacement and plastic straining response of the structure is simulated by geometrically and materially nonlinear finite element analysis. In a first case the limit load prediction of imperfect tubes (with ovalized cross sectional shape) under external pressure and discusses the accuracy of the pressure vessel code formula. The second case simulates the large strain cold-deforming process of a pressure vessel made from strain-hardening steel.

Raja et al., (2007) carried out an experiment to study the effect of various parameters such as nozzle diameter, angle of inclination, jet position and jet velocity on mixing time. An increase in the nozzle diameter was found to reduce the mixing time at a given level of power consumption also improve the energy efficiency.

Patil et al. (2016), performed transient finite element analysis of balanced stiffness valve. It is used to determine the time-varying displacements, stresses, strains, and forces in valve parts. Here performance of the Balanced Stiffness Valve, i.e. movement of pressure plates observed. Pressure Plate area is exposing to the fluid flow instantaneously as the supply pressure given to pressure plate. Hence it is essential to examine time dependent dynamic response of the valve. Maximum stress developed within the permissible safety limit also deformation is sufficient for valve performance.

Khan (2010), studied the analysis of stress distributions in a horizontal pressure vessel and the saddle supports. The results obtained from a 3D finite element analysis show that the stress distribution in different parts of the saddle separately, i.e. wears web, flange and base plates. The effect of changing the load and various geometric parameters was investigated the recommendation are made for the optimal values of ratio of the distance of support from the end of the vessel to the length of the vessel also the ratio of the length of the vessel to the radius of the vessel for minimum stresses both in the pressure vessel and the saddle structure.

Nicolas et al., (2016), did the study to compare the mixing performance of three geometries of HartridgeRoughton mixers with similar dimensions and identical inlet flow rates. Mixing efficiency of three mixers is characterized by a segaration index.

Sedmaka et.al., (2016) have studied the Elastic-plastic behaviour of welded joints during loading and unloading of pressure vessels. Pressurizing of the model was done in two stages. First the model was loaded to working

pressure then held at a lower pressure for 2 hours. After unloading it was tested at total working load or water hammer load. They conclude that higher heat input to the weld zone is better. The HAZ of microalloyed steel has greater resistance to crack during load variations compared to WM. High stress levels for initiation of stable crack growth suggest the possibility that the welded structure can operate safely even in the presence of relatively large surface cracks. The integrity of heterogeneous welded joints is not affected by the presence of surface cracks because overmatching plays a protecting role, which consists in a small plastic deformation of weld metal even at high loads causing fracture of parent metal.
Andrade et.al.,(2015) have done a comparative study on methods to analyse stresses in vessel/nozzle due to external loads. A model of a nozzle without reinforcement is prepared so that comparison can be done by WRC 107, WRC 297 and FEA method. The WRC (Welding Research Council) Bulletin 107 is a parameterized procedure of stress calculation of nozzle in which the input values are dimensionless and the stress results are obtained from curves developed based on experimental data. WRC Bulletin 297 is a supplement to WRC 107 for higher diameter- thickness ratios. The stress values obtained by the three methods were close and are reliable for pressure vessels and nozzles that fit in WRC 107 and 297 procedure.
From the literature study it can observe that the design and analysis of pressure vessel can be studied. In chemical industry storage and distribution of chemical take place in vessel. Due to incoming pressure and temperature creates the deformation in vessel which result high local stress for deeper understanding of stress by combine stress analysis. Maximum stress valve is useful for predicting the failures of pressure vessel.

- The prediction of burst pressure at which crack occurs in pressure vessel by finite element analysis. Due to this crack propagation fatigue life of pressure vessel is reduces. It overcomes by FEA.
- The finite element analysis is generally used to determine the response of a components under the action of different boundary conditions.
- Nonlinear analysis of finite element method is very accurate result than a routine finite element analysis of non-routine problem.
- Combine effect of static and thermal loading conditions on pressure vessel need to be investigate.

Design

A modern chemical process involves more complex and a series of operations which must be run continuously for many months or year. It demands the equipment of exceptional robustness, ingenuity and reliability. A verity of equipment such as pressure vessel is required for storage, handling and processing of chemical. Each equipment performs specific function in some cases it can be suitably modified to perform a different function. Condition such as temperature, pressure etc. under which the equipment is expected to perform.
Industrial horizontal cylindrical pressure vessels are usually supported on twin saddle support, which is used for the purpose of carrying different kinds of products like LPG, petroleum redacts steam and other beverages. Pressure vessels are the most widespread equipment in industrial sector. More precisely vessels are the fundamental component for the industrial importance. Usually, saddles are used to support the horizontal pressure vessel. Apart from the stress due to the internal pressure inside the vessel, saddle has to carry other stresses also such as self-weight of the vessel and other atmospheric conditions. Generally, the pressure vessels are subjected to uniform internal pressure under the effect of liquid contained by it. But due to structure of pressure vessel and loading conditions, it encounters non uniform stresses over its entire structure. So, while we are designing horizontal pressure vessels the design and analysis of its saddle supports are very important step. Saddle stiffness and distances between the saddles have a major effect on the maximum stress induced in the entire structure. The length and diameter of the pressure vessel was chosen from commercial pressure vessel sizing guide.

Selection of Material

Pressure vessel is major part of equipment used in chemical industry. Selection of material for pressure vessel is depending upon different condition such as pressure, temperature and corrosion effect due to acid and alkalis. The material selection is done based on application point of view, for pressure vessel SA 240-grade 304L, (ASME – Sec-8- div. 2-part D) is mostly used as, 304L is an excellent choice for service in lower than ambient temperature application. It has excellent notch toughness and is used in both pressure vessels and industrial boilers.
In this pressure vessel during operation, maximum operating internal pressure is 1.3 Mpa and operating temperature is 77°C. Hence, it creates continuous deformation and distortion in the pressure vessel. A stress induced in a vessel beyond the allowable limit, which causes failure of the wall and other components. The failure of a vessel not only due to the static conditions but it may be due to generation of thermal stresses in vessel material during the operation. Hence, it is the need to validate design calculation with the help of FEM analysis to find out design safety and failure reasons of pressure vessels. The analysis results show the failure areas and maximum deformation of the vessel with the combined effect of loading conditions in multi-physics analysis. Hence this investigation focuses on, design and multi-physics stress analysis of pressure vessel applied in low pressure and temperature working conditions.

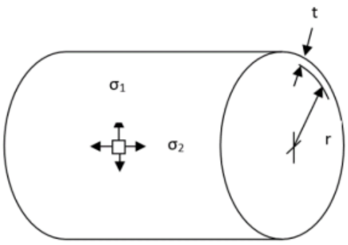

Fig 2 (a)

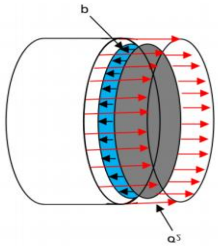

Fig 2 (b)

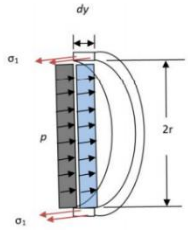

Fig 2 (c)

In general, analytical approach could be considered as the fundamental tool in solving problems related to engineering mechanics. Therefore, in this study for analytical approach the fundamental equations were used to determine the axial stress and longitudinal stress for the thin-walled pressure vessel respectively. For a cylindrical vessel, a gauge pressure p is developing within the vessel by containing gas or fluid, which is assumed to have negligible weight. Fig. 2 (a,b,c) shows the stress direction, cylindrical pressure vessel subjected to the normal stress σ1 in the circumferential or hoop direction and σ2 in the longitudinal or axial direction.

Analytical calculations are carried out using following input design parameters as per the customer requirement as shown in Table 3.

Table 3. Pressure vessel design parameters

Design pressure	1.3 Mpa
Design temperature	77°C
Vessel thickness	16 mm
Inner radius	1500 mm

Allowable stress calculation-

Allowable stress = yield strength / factor of safety
= 262/1.5
= 174 Mpa

Actual allowable stress = allowable stress X 0.85 (duty factor)
= 174 X 0.85
= 148 Mpa

Design of cylindrical shell

Allowable stress = 148 Mpa

(According to ASME Section-VIII Division-I, UG36)

Thickness of shell, t = 16 mm

Circumferential stress = σ_1= pr/t = (1.3 x 1500) / 16
= 121.875 Mpa

Longitudinal stress= σ_2 = pr/2t = (1.3 x 1500) / (2 x 16)
= 60.9375 Mpa

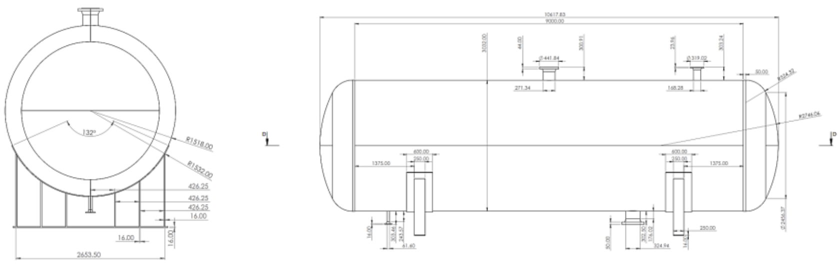

Figure 3. 2D sketch of pressure vessel

Finite Element Method

The analysis of the pressure vessel was carried for two different conditions. In the further section, detailed results and discussion were presented.

Case 1: Stress analysis under the combined loadings of pressure vessel with both saddles fixed and other thermomechanical conditions

- Thermal boundary conditions as internal temperature 77°C and convection on exterior surface as stagnant air simplified case with coefficient of convection is 5 e-6 W/ mm^2 °C
- Static boundary conditions as internal pressure 1.3 Mpa and both saddles are fixed in all degrees of

freedom as shown in Fig.11. For the pressure applied between 1.3 Mpa, and temperature 77℃ (thermomechanical analysis).

Fig.7 a) shows the CAD geometry of pressure vessel. Fig.8 b) shows the mesh model with 75530 number of nodes and 36634 number of elements.

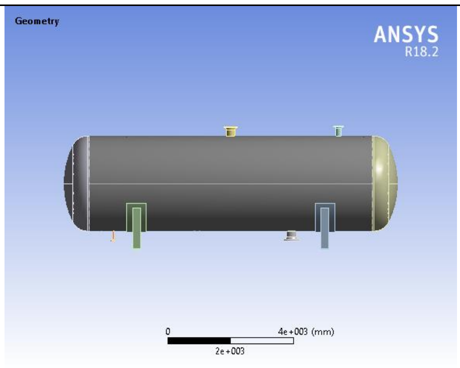

Figure 7. a) Geometry

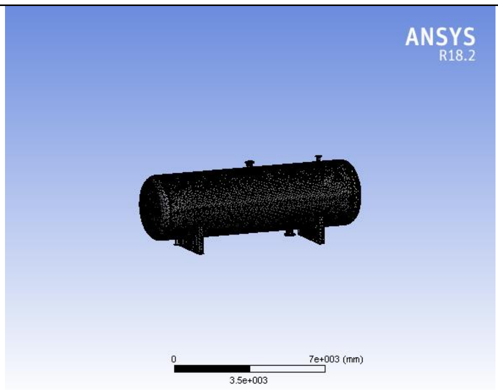

Fig.8 b) Mesh model

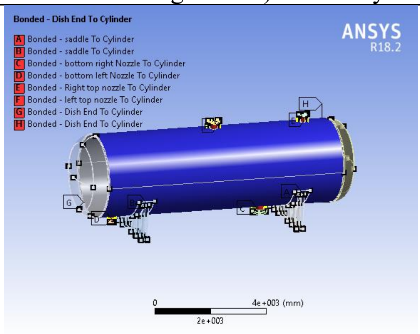

Fig.9 Contacts

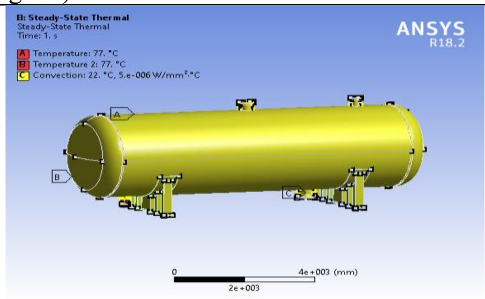

Fig.10 Thermal boundary condition

Thermal analysis

Heat transfer is defined as energy in transit. Analysis of a system using the laws of heat transfer is named as Thermal Analysis. Heat transfer is a branch of thermodynamics which deals with rate of heat transfer between two or more equilibrium states of a system. Thermal analysis is investigation of the part or a system, to calculate heat transfer rate and temperature distribution.

Static analysis

Static structural analysis determines the stress, strain and deformation of a component or assembly can be investigated under a range of load conditions to ensure that expensive failures are avoided at the design stage. Static stress analysis is arguably the most common type of structural analysis using FE method. Structural loads are typically one, or a combination of the following:

- External forces such as clamping force in subsea connectors.
- Surface loads, e.g. pressure loading in pressure vessels
- Body forces (gravity, acceleration such as centrifugal force in rotating machines)

The structural response to more complex loads, for instance those arising from thermal analysis, can also be simulated using the multi-physics approach.

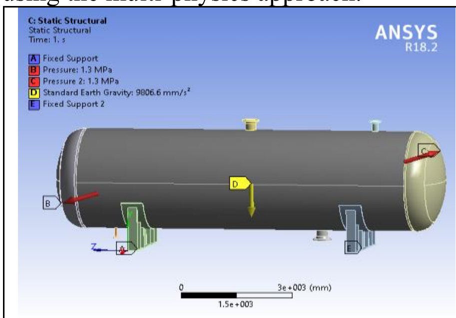

Fig.11 Static condition

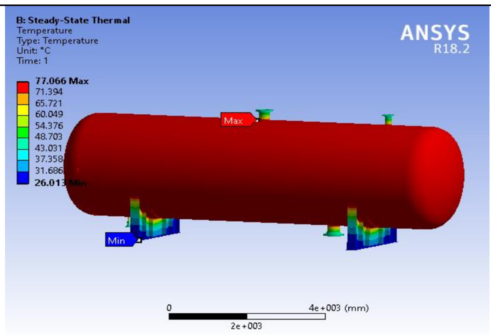

Fig.12 Temp. distribution

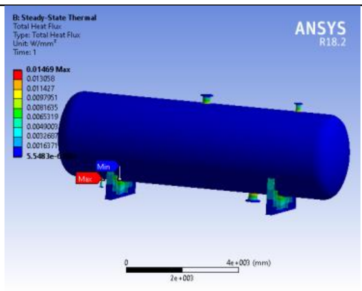

Fig.13 Total heat flux

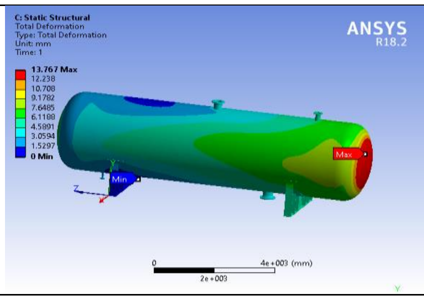

Fig.14 Total deformation

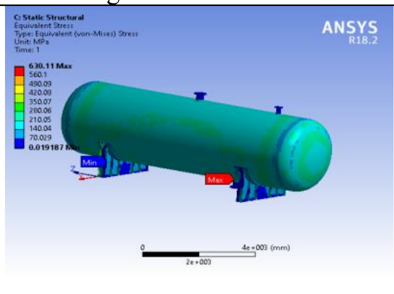

Fig.15 Equivalent stress

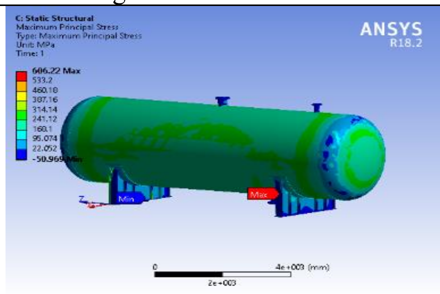

Fig.16 Max principal stress

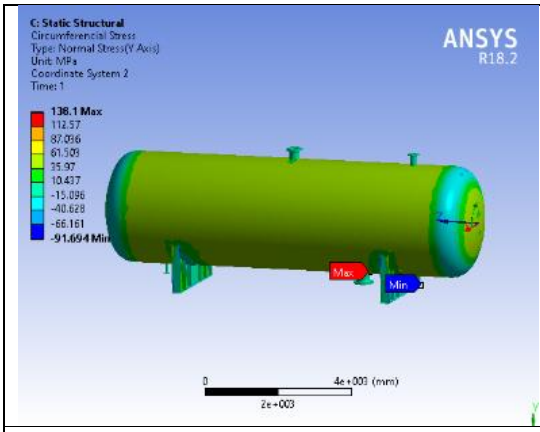

Fig.17 Circumferential stress

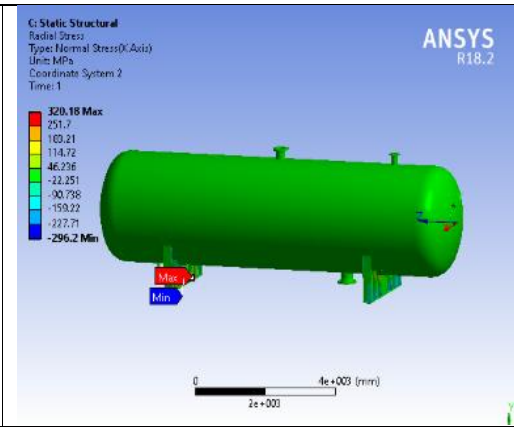

Fig.18 Radial stress

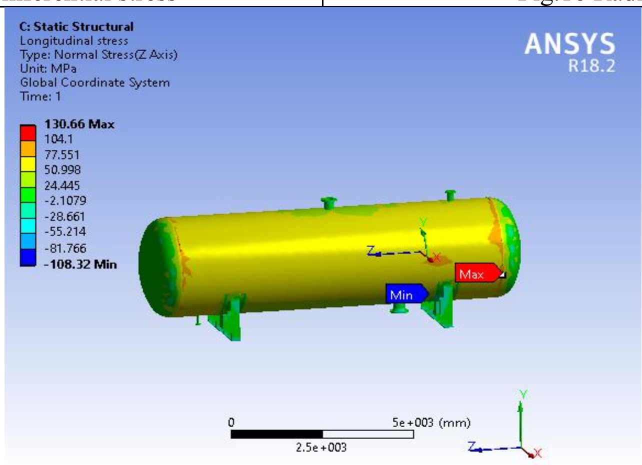

Fig.19 Longitudinal stress

Table 4 Results of FEA when both saddles fixed

Total deformation	Equivalent stress	Maximum principal stress	Circumferential stress	Longitudinal stress	Radial stress
13.767 mm	630.11 Mpa	606.22 Mpa	138.1 Mpa	130.66 Mpa	320.18Mpa

The maximum principal stress was found to be 606.22 Mpa, it is not within the permissible limit. It is observed that the current analysis results are not close to the allowable stress values. This proves that the current FE model is not accurate for saddles fixed in all degrees of freedom. The summary of all results is mentioned in Table 4. The deformation in the vessel is 13.767 mm which is higher and it may cause the failure in the vessel in longer service life. Also, all other stress values are not within the limit. So, the vessel is not acceptable for further analysis. Hence the case 2 were considered for further analysis, which is mentioned in further sections.

5.2 Case 2: Stress analysis under the combined loadings of pressure vessel with one saddle fixed and other saddle is free in Z axis (longitudinal direction)

- Thermal boundary conditions as internal temperature 77°C and convection on exterior surface as stagnant air simplified case with coefficient of convection is 5 e-6 W/ mm^2 °C shown in fig.10
- Static boundary conditions as internal pressure 1.3 Mpa and left-hand saddle is fixed and right-hand saddle is free along axial direction (displacement along Z axis) as shown in Fig.20. It is observed that the current results are very close to the allowable stress values.

Static boundary conditions

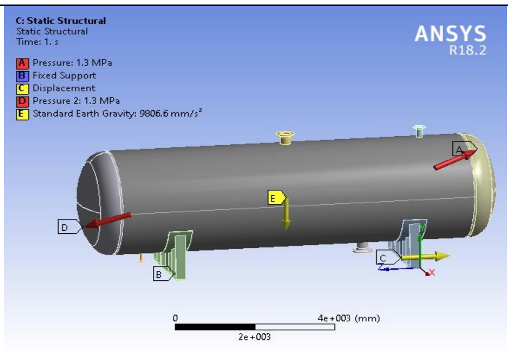

Fig.20 Static boundary condition

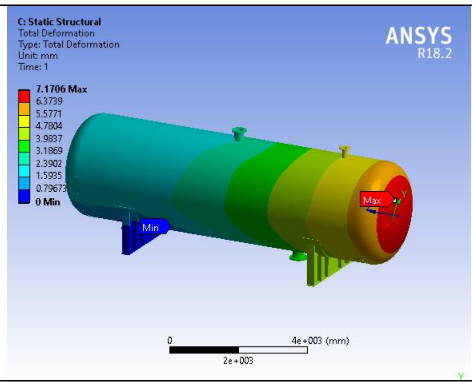

Fig.21 Total deformation

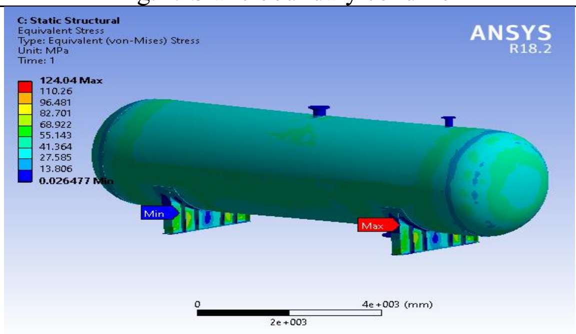

Fig.22 Equivalent stress

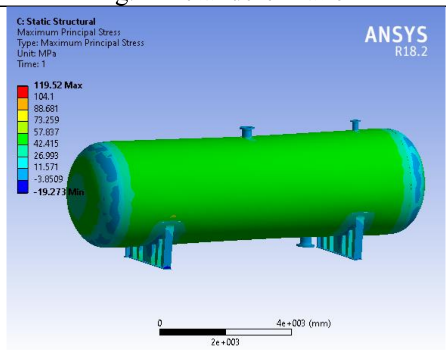

Fig.23 Max. principal stress

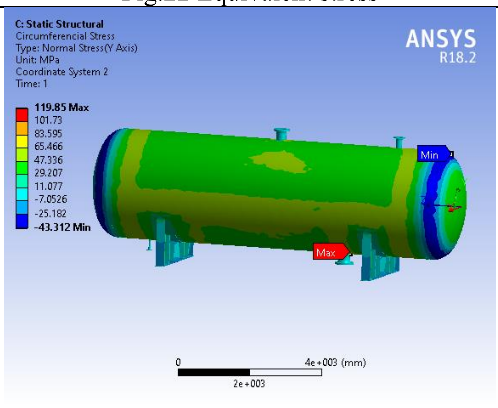

Fig.24 Circumferential stress

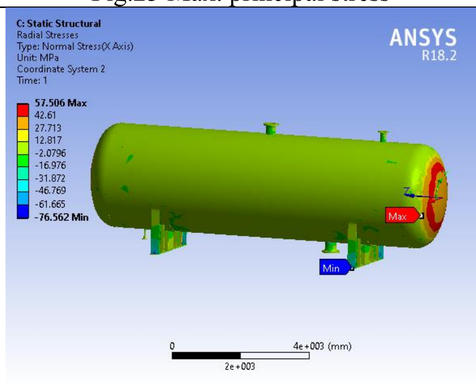

Fig.25 Radial stress

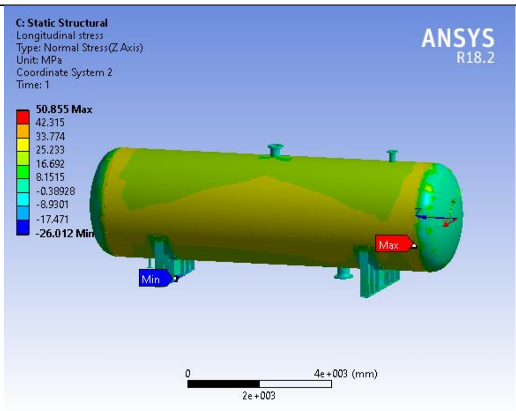

Fig.26 Longitudinal stress

For the pressure applied between 1.3 Mpa, and temperature 77°C (thermomechanical analysis) the maximum principal stress was found 119.52 Mpa, it is within the permissible limit. As analytical allowable stress is 148Mpa. Maximum principal stress less than allowable stress, this proves that the current FE model is accurate for all given boundary conditions. The summary of all results are mention in Table 5.

Table 5 Results of FEA when LSH saddle fixed and RSH is free along Z axis

Total deformation	Equivalent stress	Maximum principal stress	Circumferential stress	Longitudinal stress	Radial stress
7.1706 mm	124.04 Mpa	119.52 Mpa	119.85 Mpa	50.855 Mpa	57.50 Mpa

The deformation in the vessel is 7.1706 mm which is acceptable for given boundary conditions. The equivalent stress is found 124.04 Mpa which is in within limit. The circumferential and longitudinal stresses were found to be 119.85 Mpa and 50.855 Mpa respectively. The analytical circumferential and longitudinal values were 121.87 Mpa and 60.93 Mpa. As the analytical and FE stress values are close to each other. Hence the FE model pressure vessel was validated for given boundary conditions. This could show that the FE model in this current study and simulated results obtained is accurate and reliable. The table 6 shows the comparative values of deformation and thermal stresses for case 1 and case 2. Table 7 shows the validation of results by comparing analytical and FEA results.

Table 6 Comparative results of deformation and thermal stresses

Parameters	Case 1: Both saddles fixed	Case 2: LSH saddle fixed and RHS saddle free along axial direction	Remark
Total deformation (mm)	13.767	7.170	Case 2 acceptable
Equivalent stress (Mpa)	630.11	124.04	
Circumferential stress (Mpa)	138.1	119.85	
Longitudinal stress (Mpa)	130.66	50.855	
Radial stress (Mpa)	320.18	57.50	
Maximum principal stress (Mpa)	606.22	119.52	
Allowable stress (Mpa)	148	148	
Remark	**Not safe**	**Safe**	

Table 7 Comparison between analytical and FEA results

Parameters	Analytical	FEA	Remark
Couple field circumferential stress (Mpa)	121.87	119.85	Safe
Couple field longitudinal stress (Mpa)	60.93	50.855	Safe
Couple field maximum principal stress (Mpa)	148	119.52	Safe
Equivalent stress (Mpa)	148	124.04	Safe
Total deformation (mm)	NA	7.170	Safe

Modal Analysis

Modal analysis is a powerful tool to identify the dynamic characteristics of structures. Every structure vibrates with high amplitude of vibration at its resonant frequency. It is imperative to know the modal parameters- resonant

ISSN: 1001-4055
Vol. 46 No. 04 (2025)

frequency, mode shape and damping characteristics of the structure at its varying operating conditions for improving its strength and reliability at the design stage. The theoretical modal analysis technique has also been investigated using finite element method (FEM). Modal analysis has been performed after creating the pressure vessel finite element model and meshing in free-free state and with no constraints. The results have been calculated for the first 6 frequency modes. In this analysis we have made use of subspace method in ANSYS. 3 modes are related to the chassis displacement in x, y and z directions and 3 modes are related to chassis rotation about x, y and z axes. In Fig.27-32 related natural frequencies and mode shapes for pressure vessel with maximum displacement in each mode, have been shown. The first, modes frequency shows principal frequency called as natural frequency of vibration.

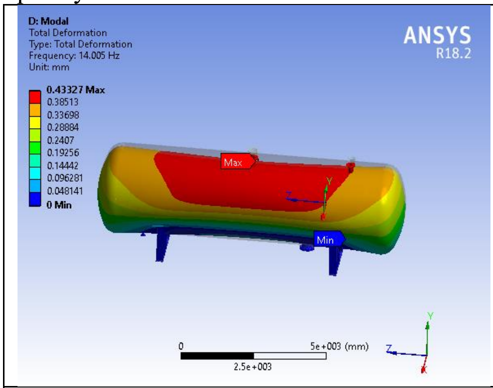

Fig. 27 Mode shape 1

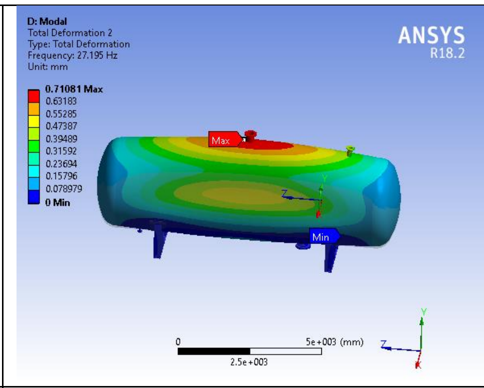

Fig. 28 Mode shape 2

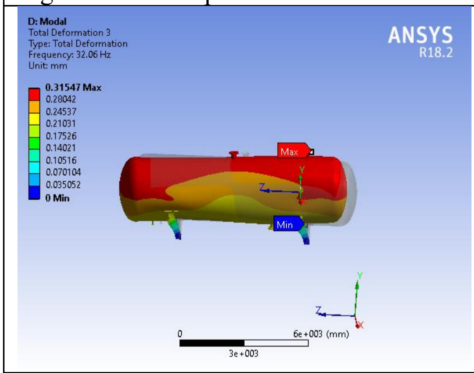

Fig. 29 Mode shape 3

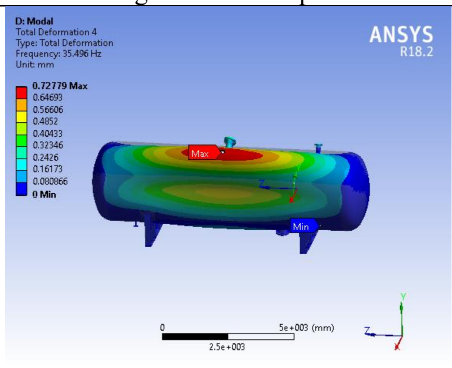

Fig. 30 Mode shape 4

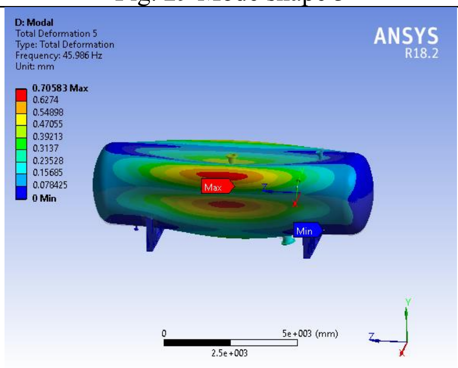

Fig. 31 Mode shape 5

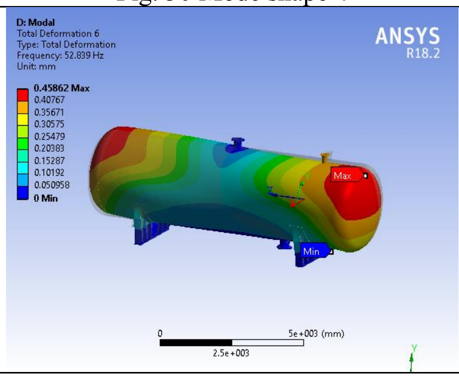

Fig. 32 Mode shape 6

Results of modal analysis

The first mode shape with 14.005 Hz frequency as a principal frequency of vibration. The second mode is a vertical bending at 27.195 Hz. At this mode, the maximum translation is at the upper part of the pressure vessel. The third and fourth modes are localized bending modes at 32.06 Hz and 35.495 Hz. The maximum translation is experienced at top side of pressure vessel. The vessel also experienced big translation at fifth mode which is a localized mode. The sixth mode is the torsion mode at 52.839 Hz with maximum translation at both ends of the chassis. Found natural frequencies from modal analysis of pressure vessel, are used for determining the suitable situations for vessel parts in working conditions.

Table 8 Mode shapes and corresponding natural frequency

Mode No	Natural frequency
1	14.005
2	27.195
3	32.06
4	35.496
5	45.986
6	52.839

Conclusion

The design of the pressure vessel was carried out as per the ASME boiler and pressure vessel code. In the first case, when the right-hand saddle was fixed in all degrees of freedom, combined static and thermal stresses and deformations were not within acceptable limits. However, when the right-hand saddle was free along longitudinal / axial direction, the combined static and thermal stresses were within acceptable limits. In both cases the left-hand side saddle was kept fixed. Software and mathematical approach show that stress calculated by ANSYS is less than that by mathematical approach and also less than allowable stresses.

- The equivalent thermal stresses due to combined loading of static and thermal boundary conditions in case 2 are obtained as 124.04 Mpa. These values are acceptable as compared to the yield strength of the vessel material.
- The analytical values of circumferential and longitudinal stress stresses are 121.87 Mpa and 60.93 Mpa respectively, and the ANSYS values are obtained in the case 2 as 119.85 Mpa and 50.855 Mpa thus both stresses are within acceptable limit as per ASME boiler and pressure vessel standard.
- The paper has looked into the determination of the dynamic characteristic the natural frequencies and the mode shapes of the truck chassis, investigating the mounting locations of components on the truck chassis and observing the response of the truck chassis under static loading conditions. The first six natural frequencies of the pressure vessel are below 100 Hz and vary from 14.005 Hz to 52.839 Hz. For the first four modes, the vessel experienced global vibration except for the fifth mode. The global vibrations of the vessel include torsion, lateral bending and vertical bending with 2 and 3 nodal points.

Table 7 shows the comparative results of stresses using analytical and FEA approach for case 2. Table 8 shows the frequency of vibration results for determination of natural frequency of vibration. It has been found that the case 2 design is safe.

References

1. Adithya, M. and Patnaik, M.M.M., 2013. Finite Element Analysis of Horizontal Reactor Pressure Vessel Supported on Saddles. *International Journal of Innovative Research in Science, Engineering and Technology*, *2*(7), pp.3213-3220.
2. Apurva Pendbhaje, Mahesh Gaikwad, Nitin Deshmukh and Rajkumar Patil, 2014. Design and Analysis of Pressure Vessel. *International journal of innovative research in Science and Technology*, 2(3): 28-34.
3. ASME, ASME Boiler & Pressure Vessel Code Section VIII Division 1 New York, 1998.*American Society of Mechanical Engineers,* (Cited in Zengliang Gao., etal, 2005)
4. ASME, Section II Part D New York, 1998. *American Society of Mechanical Engineers.* (Cited in Zengliang Gao., etal, 2005)
5. Deolia, P. and Shaikh, F.A., 2016. Finite element analysis to estimate burst pressure of mild steel pressure vessel using Ramberg–Osgood model. *Perspectives in Science*, *8*, pp.733-735.
6. Gedam, A.C. and Bhope, D.V., 2015. Stress analysis of pressure vessel with different end connections. *Int. J. Mech. Eng*, *3*(11), pp.19-27.
7. Gupta, S.R., Desai, A. and Vora, C.P., 2014. Optimize nozzle location for minimization of stress in pressure vessel. *IJIRST-International Journal for Innovative Research in Science and Technology*, *1*(1).
8. Huda, Z. and Ajani, M.H., 2015. Evaluation of Longitudinal and Hoop Stresses and a Critical Study of Factor of Safety (FoS) in Design of a Glass-Fiber Pressure Vessel. *International Journal of Civil, Environmental, Structural, Construction and Architectural Engineering*, *9*(1), pp.39-42.
9. K. Patil and V. Patil, 2016. Analysis of balance stiffness valve by using transient analysis method. *International Journal on Recent and Innovation Trends in Computing and Communication, vol*, 7, pp. 145-151.
10. Kavekar, M.M., Khatawate, M.V.H. and Patil, M.G.V., 2013. Weight reduction of pressure vessel using FRP composite material. *International Journal of Mechanical Engineering and Technology (IJMET)*, *4*(4), pp.300-310.
11. Khan, S.M., 2010. Stress distributions in a horizontal pressure vessel and the saddle supports. *International Journal of Pressure Vessels and Piping*, *87*(5), pp.239-244.

12. Kong, F. and Kim, H.D., 2014. *Starting transient simulation of a vacuum ejector-diffuser system under Chevron effects. International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics.*
13. Lu, Y.Q. and Hui, H., 2015. Investigation on mechanical behaviors of cold stretched and cryogenic stretched austenitic stainless steel pressure vessels. *Procedia Engineering*, *130*, pp.628-637.
14. Mundhe, N.D. and Utpat Abhay, A., 2013. Analysis of cracked cylindrical pressure vessel by using experimental approach. *International Journal of Latest Trends in Engineering and Technology (IJLTET), vol*, *2*(3), pp.155-160.
15. N. Di Patrizio, M. Bagnaro, A. Gaunand, J. Hochepied, D. Horbez and P.Pitiot, 2016. *Hydrodynamics and mixing performance of Hartridge Roughton mixers: Influence of the mixing chamber design. Chemical Engineering Journal*, vol. 283, pp. 375-387.
16. Nabhani, F., Ladokun, T. and Askari, V., 2012. Reduction of stresses in cylindrical pressure vessels using finite element analysis. *INTECH Open Access Publisher.*
17. Narale, P. and Kachare, P.S., 2012. Structural analysis of nozzle attachment on pressure vessel design. *International Journal of Engineering Research and Applications (IJERA) ISSN*, pp.2248-9622.
18. Narasaiha, L. (2009). Finite element analysis. CRC Press.
19. Pande, S.S., Darade, P.D. and Gogate, G., 2014. Transient analysis and fatigue life prediction of tubesheet. *International Journal of Research in Engineering and Technology*, *3*(09), pp.464-471.
20. Patel, T.M., Bhatt, D.M. and Patel, H.K., 2013. Analysis and validation of Eicher 11.10 chassis frame using Ansys. *International Journal of Emerging Trends & Technology in Computer Science*, *2*(2), pp.85-88.
21. Porter, M.A., Martens, D.H. and Marcal, P., 1999. On using finite element analysis for pressure vessel design. *ASME-PUBLICATIONS-PVP*, *388*, pp.191-196.
22. Prasanna, P., Sushma, J. and Srikanth, K.P., 2018. Stress analysis and design optimization of a pressure vessel using Ansys package. *Materials Today: Proceedings*, *5*(2), pp.4551-4562.
23. Qadir, M. and Redekop, D., 2009. SCF analysis of a pressurized vessel–nozzle intersection with wall thinning damage. *International journal of pressure vessels and piping*, *86*(8), pp.541-549.
24. Rafiee, R. and Torabi, M. (2018). Stochastic prediction of burst pressure in composite pressure vessels. Composite Structures, 185, pp.573-583.
25. Raparla, S.K. and Seshaiah, T., 2012. Design and analysis of multilayer high pressure vessels. *International Journal of Engineering Research and Application*, *2*(1), pp.355-361.
26. Saidpatil, V.V. and Thakare, A.S., 2014. Design & weight optimization of pressure vessel due to thickness using finite element analysis. *International Journal of Emerging Engineering Research and Technology*, *2*(3), pp.1-8.
27. Sanal, Z., 2000. Nonlinear analysis of pressure vessels: some examples. *International journal of pressure vessels and piping*, *77*(12), pp.705-709.
28. Sedmak, S., Algool, M., Sedmak, A., Tatic, U. and Dzindo, E., 2016. Elastic-plastic behaviour of welded joints during loading and unloading of pressure vessels. *Procedia Structural Integrity*, *2*, pp.3546-3553.
29. Skopinsky, V.N. and Smetankin, A.B., 2006. Modeling and stress analysis of nozzle connections in ellipsoidal heads of pressure vessels under external loading. *Applied Mechanics and Engineering*, *11*(4), p.965.
30. Sonawane, P.B. and Damle, P.G., 2016. Design And Analysis of Top Conical End Closure Nozzle Junction And Pressure Vessel. *International Journal of Engineering Sciences & Research Technology, Value*, *3*.
31. Thakkar, B.S. and Thakkar, S.A., 2012. Design of pressure vessel using ASME code, section VIII, division 1. *International Journal of Advanced Engineering Research and Studies*, *1*(2), pp.228-234.
32. Wasewar, K.L. and Sarathi, J.V., 2008. CFD modelling and simulation of jet mixed tanks. *Engineering Applications of Computational Fluid Mechanics*, *2*(2), pp.155-171.