Shailendra Ramchandra Gawand, 2025, 13:6
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science, Engineering and Technology
An Open Access Journal

Analysis aspects of pressure tank using Ansys

[1]Shailendra Ramchandra Gawand, [2]Santosh Rama Shekokar, [3]N. A. Kharche
Pad. Dr. V B Kolte College of Engg Malkapur

Abstract - This review paper presents a detailed analysis of pressure vessels, focusing on finite element analysis (FEA) to evaluate structural integrity. Pressure vessels are widely used in industries such as chemical processing, power plants, and oil refineries, where they are subjected to extreme temperature and pressure conditions. Ensuring their safe and efficient operation requires a thorough understanding of stress distribution, deformation characteristics, and failure mechanisms. This study reviews existing research on the thermal stresses and deformations that develop in pressure vessels under various loading conditions. The analysis compares results from FEA with analytical methods to assess the accuracy and reliability of computational approaches. The pressure vessel is designed according to ASME Section VIII Division 2 standards, and key parameters such as shell thickness, head dimensions, and structural stability are evaluated using simulation tools. To gain deeper insights into structural performance, static and thermal analyses are conducted using ANSYS, incorporating multi-physics simulations that consider combined loading effects. Additionally, modal analysis is reviewed to determine the natural frequency of pressure vessels, which is crucial for understanding vibration characteristics and avoiding resonance-related failures. A specific focus is given to the effects of different constraint conditions, particularly those related to saddle support configurations. The study reviews two different constraint conditions of the right-hand saddle, examining their influence on stress distribution and deformation patterns under constant internal design temperature and pressure. This comparative assessment provides valuable insights into optimizing support structures to enhance pressure vessel performance and longevity. Overall, this review consolidates key findings from previous studies on pressure vessel analysis, highlighting advancements in FEA methodologies and their application in structural assessment. The insights presented in this paper aim to aid engineers and researchers in designing more reliable pressure vessels with improved safety margins and operational efficiency.

Keywords - Reactors, Finite element analysis (FEA), Stress analysis.

I. INTRODUCTION

Pressure vessels are critical containment systems designed to operate at pressures significantly different from ambient conditions [1]. These engineered structures present substantial safety challenges, as operational failures can lead to catastrophic consequences [2]. This study focuses on the analysis of pressurized oil tanks, with particular emphasis on identifying key parameters that govern operational efficiency and structural integrity under varying service conditions. Modern pressure vessel design follows rigorous standards, particularly ASME Boiler and Pressure Vessel Code Section VIII [3], which ensures safety through comprehensive engineering specifications. These containers find extensive application across industries including nuclear power generation, petroleum refining, and chemical processing [4]. The growing demand for alternative fuels has further increased requirements for vessels capable of withstanding extreme pressures and temperatures [5]. Recent advancements in pressure vessel technology include:

Development of advanced materials and composites [6]

Improved welding techniques [7-9]

Enhanced computational analysis methods [10]

Finite element analysis (FEA) has become indispensable for understanding complex phenomena such as fatigue behavior and creep deformation [11]. Multi-physics modeling enables comprehensive evaluation of combined mechanical and thermal loads [12], while transient dynamic analysis provides insights into time-dependent stress distributions [13]. These computational tools allow for accurate prediction of vessel performance under operational conditions, significantly improving design reliability.

II. LITERATURE REVIEW

Design Standards and Analysis Methods

The ASME standards provide a robust framework for pressure vessel design, significantly reducing development time while ensuring safety [1]. Finite element analysis has emerged as a critical tool for evaluating vessel configurations, particularly for saddle-supported designs [14]. Studies have demonstrated the effectiveness of FEA in optimizing wall thickness and temperature distribution in boiler applications [15].

Nonlinear FEA techniques enable accurate simulation of large displacements and plastic deformation in pressure vessel components [16]. These advanced methods provide valuable insights into limit load predictions and material behavior under extreme conditions [16].

Material Innovations

Significant progress has been made in material technology for pressure vessels: Cryogenic treatment of austenitic stainless steel can double allowable stresses, enabling 60-75% reduction in wall thickness [17]

Fiber-reinforced polymer (FRP) composites offer 75% weight reduction compared to steel while eliminating corrosion concerns [18]

Multilayered vessel designs demonstrate 26% material savings and more uniform stress distribution (12.5% variation vs 17.35% in monoblock designs) [19]

Welding and Joint Integrity

Recent research on welded joints reveals that increased heat input improves crack resistance during pressure cycling [20]. The heat-affected zone (HAZ) of microalloyed steels shows particularly good performance under variable loading conditions [20]. Advanced welding techniques, including Plasma Transferred Arc Welding (PTAW), have proven effective for applying wear-resistant coatings [7-9].

Nozzle and Mixing Dynamics

Nozzle design significantly affects mixing efficiency:

Larger nozzle diameters reduce mixing time while improving energy efficiency [21]

Hartridge-Roughton mixers achieve complete homogenization in under 5 milliseconds [22]

Optimal fuel oil-to-steam pressure ratios minimize Sauter Mean Diameter (SMD) in atomization processes [23]

Computational fluid dynamics (CFD) simulations have become essential for analyzing mixing phenomena and validating nozzle designs [24].

Structural Analysis and Failure Prevention

Comprehensive studies have investigated:

Stress distribution in saddle-supported vessels [25]

Burst pressure prediction using Ramberg-Osgood models [26]

Nozzle-vessel interactions under wall thinning conditions [27]

Transient analysis of valve components [28], The WRC Bulletin 107/297 methods provide reliable stress analysis for nozzle reinforcements, with good correlation to FEA results [29]. Ellipsoidal and dished end configurations demonstrate superior stress distribution compared to flat or hemispherical designs [30]. The thresher blade press tool was designed and analyzed, revealing that the fixture ensures proper part orientation and positioning in the assembly [31]. This review synthesizes current knowledge across these critical areas, providing a foundation for improved pressure vessel design and analysis methodologies. The integration of advanced materials, optimized geometries, and sophisticated analysis techniques continues to push the boundaries of pressure vessel performance and reliability.

Inferences from Literature Survey

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.

III. METHODOLOGY

One of the most common examples of engineering analysis is finite element analysis or FEA. FEA is one of the most commonly used and powerful features of the CAD software. To carry out the analysis of object by using FEA, the object is divided into finite number of small elements of shapes like rectangular or triangular. These objects form the interconnected network of the concentrated nodes. Analysis is being carried out by using ANSYS software. Finite Element Analysis is a mathematical representation of a physical system comprising apart/assembly (model), material properties, and applicable boundary conditions (collectively referred to as (pre-processing), the solution of that mathematical representation (solving), and the study of results of that solution (post processing). Finite element analysis (FEA) involves solution of engineering problems using computers. Engineering structures that have complex geometry and loads, are either very difficult to analyze or have no theoretical solution. However, in FEA, a structure of this type can be easily analyzed. FEA software provides a complete solution including deflections, stresses, reactions, etc. Use of proper techniques for modeling a structure, the boundary conditions and, the limitations of the procedure, are very crucial. Engineering structures, e.g., bridge, aircraft wing, high-rise buildings, mechanical component etc., are examples of complex structures that are extremely difficult to analyze by classical theory. But FEA technique facilitates an easier and a more accurate analysis. In this technique the structure is divided into very small but finite size elements (hence the name Finite Element Analysis) and that method to divide the structure into small element is called as Discretization Method.

FEA is used to determine following structural properties

- Static displacement and static stress.
- Natural frequencies and mode shapes.
- Forced harmonic response amplitude and dynamic stress
- Transient dynamic response and transient stress.
- Random forced response, random dynamic stress.

FEA requires three steps for finding deflections and stresses in a structure.

- Pre-process or modelling the structure.
- Analysis.
- Post processing.

FEM steps

Step1: Pre-processing

The pre-processing involves modeling of the structure, selection of element type, meshing of the body, material properties, applying the boundary conditions, and applying the loads. The FEA model consists of several elements that collectively represent the entire structure. In the pre-processor phase, along with the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called "mesh"

Step 2: Analysis

The processing involves generation of element stiffness matrices and global stiffness matrix, solution of simultaneous equation, determination of nodal displacement, stresses and strains. The form of the individual equations, as well as the structural equation is always,

{F} = [K]{u}

Where,

{F} = External force matrix.

[K] = Global stiffness matrix

{u} = Displacement matrix.

The equation is then solved for deflections. Using the deflection values, strain, stress, and reactions are calculated. All the results are stored and can be used to create graphic plots and charts in the post analysis

Step 3: Post processing

This is the last step in a finite element analysis. Results obtained in step 2 are usually in the form of raw data and difficult to interpret. In post analysis, a CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation, etc. A graphical representation of the results is very useful in understanding behavior of the structure.

Structural analysis is the most common application of the Finite Element Method (FEM). The structural analysis is a mathematical algorithm process by which the response of a structure to specified loads and actions is determined. This response is measured by determining the internal forces or stress resultants and displacements or deformations throughout the structure.

Types of Structural Analysis

Static Analysis:

This analysis used to determine displacements, stresses, etc. under static loading conditions. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Modal Analysis:

It is used to calculate the natural frequencies and mode shapes of a structure.

Harmonic Analysis:

It is used to determine the response of a structure to harmonically time-varying loads.

Transient Dynamic Analysis:

It is used to determine the response of a structure to arbitrarily time-varying loads.

Spectrum Analysis:

An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum.

Buckling Analysis:

Used to calculate the buckling loads and determine the buckling mode shape. Both linear (Eigen value) buckling and nonlinear buckling analyses are possible.

In structural analysis using structural loads to produce a variety of structural results. By default, 3D structural simulations are performed. Static used when loads are constant for individual sets of results. Typical application includes determining safety factors, stresses and deformation for a body or assembly under structural loading.

IV. CONCLUSIONS

The present study highlights the critical role of transient dynamic stress analysis and finite element analysis (FEA) in improving the fatigue life of mixing tanks used in chemical industries. Given the extreme operational conditions varying pressures, fluctuating temperatures, and time-dependent loading these pressure vessels experience significant stress concentrations that can lead to premature failure. Through FEA simulations in ANSYS, this research identifies key stress points, predicts fatigue behavior, and proposes design modifications to enhance structural resilience. Optimizing the nozzle angle for better mixing efficiency further contributes to improved performance and longevity. The findings underscore the importance of advanced materials, precise design adjustments, and computational analysis in developing robust pressure vessels, ensuring increased safety and reliability in industrial applications.

REFERENCES

1. Adithya, M., & 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), 3213-3220.
2. Pendbhaje, A., Gaikwad, M., Deshmukh, N., & Patil, R. (2014). Design and analysis of pressure vessel. International Journal of Innovative Research in Science and Technology, 2(3), 28-34.
3. American Society of Mechanical Engineers. (1998). ASME boiler & pressure vessel code section VIII division 1. New York: American Society of Mechanical Engineers. (Cited in Zengliang Gao, et al., 2005).
4. American Society of Mechanical Engineers. (1998). ASME section II part D. New York: American Society of Mechanical Engineers. (Cited in Zengliang Gao, et al., 2005).
5. Deolia, P., & Shaikh, F. A. (2016). Finite element analysis to estimate burst pressure of mild steel pressure vessel using Ramberg–Osgood model. Perspectives in Science, 8, 733-735.
6. Gedam, A. C., & Bhope, D. V. (2015). Stress analysis of pressure vessel with different end connections. International Journal of Mechanical Engineering, 3(11), 19-27.
7. Gupta, S. R., Desai, A., & 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., & 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), 39-42.
9. Patil, K., & Patil, V. (2016). Analysis of balance stiffness valve by using transient analysis method. International Journal on Recent and Innovation Trends in Computing and Communication, 7, 145-151.
10. Kavekar, M. M., Khatawate, M. V. H., & Patil, M. G. V. (2013). Weight reduction of pressure vessel using FRP composite material. International Journal of Mechanical Engineering and Technology (IJMET), 4(4), 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), 239-244.
12. Kong, F., & 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., & Hui, H. (2015). Investigation on mechanical behaviors of cold stretched and cryogenic stretched austenitic stainless steel pressure vessels. Procedia Engineering, 130, 628-637.
14. Mundhe, N. D., & Utpat, A. A. (2013). Analysis of cracked cylindrical pressure vessel by using experimental approach. International Journal of Latest Trends in Engineering and Technology (IJLTET), 2(3), 155-160.
15. Di Patrizio, N., Bagnaro, M., Gaunand, A., Hochepied, J., Horbez, D., & Pitiot, P. (2016). Hydrodynamics and mixing performance of Hartridge Roughton mixers: Influence of the mixing chamber design. Chemical Engineering Journal, 283, 375-387.
16. Nabhani, F., Ladokun, T., & Askari, V. (2012). Reduction of stresses in cylindrical pressure vessels using finite element analysis. INTECH Open Access Publisher.
17. Deshmukh, D. D., & Kalyankar, V. D. (2018). Recent status of overlay by plasma transferred arc welding technique. International Journal of Materials and Product Technology, 56(1-2), 23-83.
18. Deshmukh, D. D., & Kalyankar, V. D. (2019). Deposition Characteristics of Multitrack Overlay by Plasma Transferred Arc Welding on SS316Lwith Co-Cr Based Alloy–Influence of Process Parameters. High Temperature Materials and Processes, 38(2019), 248-263.
19. Deshmukh, D. D., & Kalyankar, V. D. (2021). Analysis of deposition efficiency and distortion during multitrack overlay by plasma transferred arc welding of Co–Cr alloy on 316L stainless steel. Journal of Advanced Manufacturing Systems, 20(04), 705-728.
20. Chavan, H. A., & Deshmukh, D. D. (2021). Productivity Improvement of Automotive Rod Forming Process by Simultaneous Operation Using FE Simulation. International Journal of

Modern Manufacturing Technologies (IJMMT), 13(1).

21. Deshmukh, D. D., Hyalij, N. S., & Kakade, S. P. (2021). Investigations on premature failure of 9Cr-Mo elbow during operation applied in power industry. Materials Today: Proceedings, 44, 4843-4846.
22. Kalyankar, V., Bhoskar, A., Deshmukh, D., & Patil, S. (2022). On the performance of metallurgical behaviour of Stellite 6 cladding deposited on SS316L substrate with PTAW process. Canadian Metallurgical Quarterly, 61(2), 130-144.
23. Bhoskar, A., Kalyankar, V., & Deshmukh, D. (2023). Metallurgical characterisation of multi-track Stellite 6 coating on SS316L substrate. Canadian Metallurgical Quarterly, 62(4), 665-677.
24. Kakade, S. P., Thakur, A. G., Deshmukh, D. D., & Patil, S. B. (2023). Experimental investigations and optimisation of Ni-Cr-B-Si hardfacing characteristics deposited by PTAW process on SS 410 using response surface method. Advances in Materials and Processing Technologies, 9(3), 826-842.
25. Deshmukh, D. D., & Kharche, Y. (2023). Influence of processing conditions on the tensile strength and failure pattern of resistance spot welded SS 316L sheet joint. International Journal on Interactive Design and Manufacturing (IJIDeM), 1-13.
26. Bhoskar, A., Kalyankar, V., & Deshmukh, D. (2024). Implications of FCC and HCP cobalt phases on wear performance of weld deposited cobalt-based coating. Results in Surfaces and Interfaces, 100247.
27. Deshmukh, D., Kakade, S., Kabudke, P., Mhaske, M., & Dond, D. (2024). Investigations on Hardfacing Characteristics of Ni-Based Alloy on Ass 304 Using Plasma Transferred Arc Welding at Different Speeds. International Journal of Modern Manufacturing Technologies (IJMMT), 16(2).
28. Shekokar, S. R., Chavan, H. A., Choudhari, R. M., Kharche, N. A., Kharche, Y. A., & Deshmukh, D. D. (2024). Investigation in the Influence of the Composition of Metal Ferrite Based Composites on the Dielectric Constant Through Response Surface Methodology. Journal of The Institution of Engineers (India): Series D, 1-9.
29. Wagh, S. V., Jagtap, S. G., Sable, M. J., & Deshmukh, D. (2025). Metallurgical Characterization of Cr2O3 and Al2O3 Coatings on SS 316L: Implications for Durability in Desalination Environments. Transactions of the Indian Institute of Metals, 78(3), 1-14.
30. Bhoskar, A., Naik, H., Kalyankar, V., & Deshmukh, D. (2025). Nickel-based metallurgical coating architectures for superior wear resistance in high-temperature P91 steel applications. Journal of Alloys and Metallurgical Systems, 9, 100151.
31. Chavan, H. A., & Wani, V. P. (2019). Design of combination tool for an automotive component with process optimization in metal forming. International Journal on Interactive Design and Manufacturing (IJIDeM), 13(1), 401-412.
32. Kalyankar, V. D., & Deshmukh, D. D. (2017, December). Failure investigations of failed valve plug SS410 steel due to cracking. In IOP Conference Series: Materials Science and Engineering (Vol. 282, No. 1, p. 012007). IOP Publishing.
33. Naik, H. V., Deshmukh, D. D., & Kalyankar, V. D. (2019). Study of heat treatment effect on microstructure of PTA weld deposited surface of SS 316L steel. In Advances in Additive Manufacturing and Joining: Proceedings of AIMTDR 2018 (pp. 597-607). Singapore: Springer Singapore.
34. Deshmukh, D. D., & Kalyankar, V. D. (2019). Evaluation of surface characteristics of PTAW Hardfacing based on energy and powder supplied. In Advances in Micro and Nano Manufacturing and Surface Engineering: Proceedings of AIMTDR 2018 (pp. 547-558). Singapore: Springer Singapore.
35. Chaudhari, A. Y., & Deshmukh, D. D. (2020, March). Metallurgical investigations on corrosion behavior of simple and heat treated duplex stainless steel 2205 exposed to corrosive media. In IOP Conference Series: Materials Science and Engineering (Vol. 810, No. 1, p. 012048). IOP Publishing.
36. Kakade, S., Thakur, A., Patil, S., & Deshmukh, D. (2023). Experimental evaluation and correlation of plasma transferred arc welding parameters

with hardfacing defects. In Recent Advances in Material, Manufacturing, and Machine Learning (pp. 326-331). CRC Press.

37. Bhople, N., Sapkal, N., Malwe, P., Deshmukh, D., Nikam, M., & Notla, S. (2024). A review on effects of cryotreatment on tungsten carbide tools and its expected outcomes in micromachining. Recent Advances in Material, Manufacturing, and Machine Learning, 398-405.
38. Kakade, S., & Thakur, A. (2021). Comparative Analysis And Investigations Of Welding Processes Applied For Hardfacing Using AHP. International Journal of Modern Manufacturing Technologies (IJMMT), 13(1).
39. Bhoskar, A., Deshmukh, D., Kakade, S., & Kalyankar, V. (2025). Analysis of a plug stem failure in a pneumatic control valve. Canadian Metallurgical Quarterly, 1-9.