NOVATEUR PUBLICATIONS
JournalNX- A Multidisciplinary Peer Reviewed Journal
ISSN No: 2581 - 4230
VOLUME 10, ISSUE 5, May - 2024

REVIEW ON STRUCTURAL INTEGRITY OF A PRESSURE VESSEL VIA ANSYS-BASED STRESS ANALYSIS

Asif Shaikh [1],
Prof. Surwase S. S.[2],
Prof. Doijode S. N. [3] ,
Prof. Ovarikar G.P.[4]
[1]PG student, Mechanical Engineering,
STB College of Engineering, Tuljapur,Maharashtra,India
[2]Assistant Professor, Dept.t of Mechanical Engineering,
STB College of Engineering ,Tuljapur,Maharashtra,India
[3]Assistant Professor, Dept. of Mechanical Engineering,
Terna Engineering college,Nerul,Maharashtra, India
[4]Assistant Professor, Dept. of Mechanical Engineering,
Terna Engineering College,Nerul, Maharashtra, India

ABSTRACT:
The pressure vessel is a crucial component in many industrial plants, requiring thorough stress analysis to ensure safety and reliability. These vessels face primary stresses from internal pressure and secondary stresses due to thermal loading, especially in those containing hot fluids. For example, liquid metal reactors (LMRs) undergo complex thermo-mechanical loadings. Traditional analytical methods, such as those prescribed by ASME codes, are typically used to calculate these induced stresses. However, in this project, a coupled field study using ANSYS software is conducted to gain a more comprehensive understanding of thermo-mechanical stresses. Comparing the results from ANSYS with conventional analytical techniques highlights the usefulness of commercial finite element analysis (FEA) tools in designing pressure vessels. Although FEA is not yet widely adopted in the industry for such analyses, this project demonstrates its potential advantages, particularly in managing coupled field analysis involving both thermal and mechanical Load.

INTRODUCTION

A pressure vessel is a closed container designed to hold gases or liquids at a pressure significantly different from the ambient pressure. These cylindrical vessels are commonly used to store fluids in various industries, often at elevated temperatures and in a pressurized state.

Large pressure vessels were first developed during the industrial revolution in Great Britain to serve as boilers for generating steam to power steam engines. The need for design and testing standards emerged after several catastrophic explosions caused significant loss of life, leading to the establishment of certification systems.

In an early effort to create a tank capable of withstanding pressures up to 10,000 psi (69 MPa), a 6-inch (150 mm) diameter tank was developed in 1919. This tank was spirally wound with two layers of high tensile strength steel wire to prevent sidewall rupture, and the end caps were reinforced longitudinally with high-tensile rods.

The pressure difference in these vessels is hazardous, and numerous fatal accidents have occurred throughout their development and operation. Consequently, their design, manufacture, and operation are strictly regulated by engineering authorities and enforced by laws.

Pressure vessels are used in a variety of applications in both industrial and private sectors. They can be found as industrial compressed air receivers, domestic hot water storage tanks, diving cylinders, recompression chambers, distillation towers, autoclaves, and various other vessels in mining, oil refineries, petrochemical plants, nuclear reactors, spaceship habitats, submarine habitats, pneumatic reservoirs, hydraulic reservoirs under pressure, rail vehicle air brake reservoirs, road vehicle air brake reservoirs, and storage vessels for liquefied gases such as ammonia, chlorine, propane, butane, and LPG.

One significant application of pressure vessels is in the nuclear industry. Various types of nuclear reactors are in use, and special care must be taken when designing reactor pressure vessels. Specific guidelines are established for these vessels. A liquid metal-cooled nuclear reactor (LMR) is an advanced type of reactor where the primary coolant is a liquid metal. Initially adapted for nuclear submarine use, liquid metal-cooled reactors have also been extensively studied for power generation. These reactors have safety advantages because they do not need to be kept under pressure and allow for a much higher power density than traditional coolants. However, they also pose challenges, such as difficulties with inspection and repair of a reactor immersed in opaque molten metal, and issues related to corrosion and the production of radioactive activation products, depending on the choice of metal.

This project presents the design methodology for a pressure vessel used in an LMR, which is subjected to low pressures and relatively high temperatures.

LITERATURE REVIEW

LITERATURE REVIEW: In this section research papers are discussed related to the present work. Published papers are highlight in this section.

A. M. Javed Hyder and M. Asif [1] conducted a study to optimize the location and size of openings in pressure vessel cylinders using ANSYS. Their analysis was performed on three thick-walled cylinders with internal diameters of 20 cm, 25 cm, and 30 cm, each having a height of 30 cm and a wall thickness of 20 mm. Initially, they analyzed the pressure vessel cylinders without any holes to determine the tangential, longitudinal, radial, and von Mises stresses.

They then proceeded to optimize the hole size by creating holes with diameters of 4 mm, 8 mm, 10 mm, 12 mm, 14 mm, 16 mm, and 20 mm, located at the center of each cylinder. According to Figure 14, the optimal hole size was found to be 8 mm for the cylinder with a 20 cm internal diameter, and 10 mm for the cylinder with a 25 cm internal diameter. For the cylinder with a 30 cm internal diameter, a hole size of 12 mm resulted in the lowest von Mises stress values.

Furthermore, they examined the effect of hole location by placing 12 mm holes at various positions along the cylinder height: 1/6, 1/8, 2/8, 3/8, and 4/8 from the top. They observed that the von Mises stress was highest at the center (0.500 height) and decreased as the hole location moved away from the center. However, the stress increased when the hole location changed from 0.1250 to 0.0625 from the cylinder top, likely due to end effects. Ultimately, they concluded that the von Mises stress was minimized when the hole was located at 1/8 of the cylinder height.

B. Josip Kacmarcik, Nedelijko Vukojevic, and Fuad Hadzikadunic [2] conducted a study comparing stress concentration factors determined using two different methods: experimental strain gauge measurements and finite element analysis (FEA) using ABAQUS software. They investigated two different nozzle geometries, defining stress concentration factors by both maximum principal stress and maximum von Mises stress.

The study revealed that the results from both methods showed good agreement. Specifically, they examined two nozzles with different external radii—C1 nozzle having a larger radius than C2—but both having the same vessel wall thickness and external radius. Due to symmetrical properties, only 1/8 of the vessel part and 1/4 of the nozzle part were modeled using 3D tetragonal elements for mesh generation.

The stress concentration factors were determined to be 295 from both FEM analysis and strain gauge measurements. When comparing the two methods, a maximum deviation of 15.5% was observed, which is acceptable for engineering applications. This indicates that FEM analysis is reliable for determining stress concentration factors in pressure vessel design.

Furthermore, the research highlighted the advantages of FEM analysis, particularly its ability to determine stresses on the internal side of the vessel, which can be greater than external stresses and are difficult to measure with strain gauges.

C. V.N. Skopinsky and A.B. Smetankin [3] presented a study on the modeling and stress analysis of nozzle connections in the ellipsoidal head of a pressure vessel under external loading. Utilizing Timoshenko shell theory and the finite element method (FEM), they examined how external loading influences stress concentration, noting that it has a more significant impact than internal pressure. Their findings indicated an appreciable increase in maximum stress within the shell's interaction region, even at low nominal stress levels.

The study highlighted that non-radial and offset nozzle connections exhibit a non-uniform stress distribution along the interaction curve between the nozzle and the vessel head. The influence of the angular parameter α for non-radial nozzle connections was particularly notable. They observed that increasing the angle α significantly reduces the maximum effective stress for non-central connections. However, in cases of torsional moment loading, the angle α affected the stress in the opposite manner, with the stress in the shell increasing as the angle α increased. This detailed analysis underscores the complex relationship between nozzle positioning and stress distribution in pressure vessel design.

D. J. Fang and Q.H. Tang [4] conducted a comparative study on the strength behavior of cylindrical shells interacting with and without pad reinforcement under out-of-plane moment loading on the nozzle. For their research, they designed and fabricated three pairs of full-scale test vessels with varying mean diameter ratios between the nozzle and the cylindrical vessel. The materials used for the cylinder, reinforcement pad, and nozzle were low carbon steel.

Their findings revealed that pad reinforcement significantly reduces the maximum elastic stress and stress ratio, with reduction rates observed to be between 20-60% in tests and 28-59% in finite element analysis. The reduction rate depends on the structure and dimensions of the vessel, such as the D/d ratio (mean diameter of the nozzle to mean diameter of the cylindrical vessel). Additionally, the study indicated that pad reinforcement increases the plastic limit of the nozzle in the cylindrical vessel, with an increase rate of approximately 40-70% from tests and more than 40% from finite element analysis.

Overall, the results conclude that reinforcement structures are effective in enhancing the strength of pressure vessels under static external loads on the nozzle, demonstrating their practical utility in such applications.

E. Pravin Naral and P.S. Kachare [5] presented a study on the structural analysis of nozzle attachments on pressure vessel design. They noted that placing a nozzle on the top of the dished end of a vessel does not disturb the vessel's symmetry, whereas placing it on the periphery can disrupt symmetry. The size, diameter, angle, and other factors of nozzle connections can vary significantly even within a single pressure vessel, leading to geometric discontinuities in the vessel wall and causing stress concentrations around the openings. High stress concentrations at these junctions can lead to failure, necessitating detailed analysis.

In their study, they investigated the effects of nozzle angle and the number of nozzles on the periphery of a pressure vessel to achieve symmetry and maintain stress within acceptable limits. Using finite element analysis (FEA) via ANSYS software, they conducted a series of tests:

Calculated stresses with a single nozzle on top of the shell.

- Placed two nozzles at a 60-degree angle from each other and calculated stresses.
- Placed two nozzles at a 90-degree angle from each other and calculated stresses.
- Placed two nozzles at a 180-degree angle from each other and calculated stresses.
- Placed three nozzles at a 60-degree angle from each other and calculated stresses.
- Placed three nozzles at a 90-degree angle from each other and calculated stresses.
- Placed four nozzles at a 60-degree angle from each other and calculated stresses.
- Placed four nozzles at a 90-degree angle from each other and calculated stresses.

The results demonstrated that symmetrical nozzle attachments resulted in lower peak stresses compared to other configurations. Specifically, the stress increment factor was lower for symmetrical nozzle attachments. The study found that the stress values were minimized when two nozzles were placed at a 180-degree angle and when four nozzles were placed at a 90-degree angle from each other. This indicates that symmetrical nozzle attachment consistently resulted in lower stress levels than asymmetrical configurations.

F. James J. Xu, Benedict C. Sun, and Bernard Koplik [6] conducted a study on the local pressure stress at the junction of lateral pipe-nozzles with various angles of interaction. This paper reports on the variation of local pressure stress factors as the nozzle angle changes from 90 to 30 degrees. The circumferential and longitudinal stresses at four symmetric points around the pipe-nozzle junction were plotted as functions of the angle. The ALGOR finite element software was used to model the true pipe-nozzle geometry. The numerical stress results were derived from parameters beta and gamma, representing the nozzle mean radius and pipe thickness, respectively.

At a 90-degree angle, the local stress values were low. As the angle of interaction decreased from 90 degrees, the stress increased, with a significant increase observed when the angle was reduced below 45 degrees. The inside crotch point B exhibited the worst circumferential stress values. The study concluded that at a 90-degree angle, the local pressure stress at points A and B were equivalent to those at points C and D due to symmetry, resulting in lower stress values compared to other angles.

G. Amran Ayob [7] performed stress analysis on a torispherical shell with a radial nozzle. In this study, experimental readings were taken using 0.0625-inch foil strain gauges bonded to the outer and inner surfaces of the shell. The model was equipped with 39 pairs of 0.0625-inch foil strain gauges, positioned between S=-0.1 to S=0.5 in the meridional direction. These experimental results were part of a test program conducted by Drabbles to determine the shakedown behavior of a torispherical vessel with a nozzle under internal pressure, thrust, and bending moment applied to the nozzle.

The study identified three geometric locations that could influence the stress field: the sphere-nozzle junction, the sphere-knuckle junction, and the cylinder-knuckle junction. The maximum stress could occur at any of these junctions. The paper presented graphs of the elastic stress factor distribution along the meridional plane for four load cases. The crotch corner and the weld-crown region were identified as the highest stress areas, with an elastic stress factor (ESF) of approximately 2.296.

H. V. N. Skopinsky[8] had worked on a more comprehensive investigation of the shell intersection problem in ellipsoidal pressure vessel heads. We delve deeper into the stress analysis of nozzle connections, particularly when the nozzle is significantly displaced from the head axis.

Key Aspects:

- **Shell Theory and Finite Element Method:** Combining these approaches provides a robust framework for analyzing stress distribution.
- **Non-Central Nozzle:** The analysis specifically considers nozzles positioned away from the head axis, a common scenario in real-world design.
- **Numerical Procedure:** The paper details the numerical procedures employed for the analysis.
- **Structural Modeling:** The complexities of modeling the nozzle-head shell intersection are discussed.
- **SAIS Program:** The paper explores the capabilities of the SAIS special-purpose computer program for stress analysis and potential optimization.
- **Stress Analysis Results:** The paper presents the findings of the stress analysis, including parametric studies.
- **Focus on Practical Design:** The analysis emphasizes the importance of understanding this under-investigated problem for improving the reliability of pressure vessel head designs.
- **Design Optimization:** The paper highlights the potential of the SAIS program for optimizing nozzle placement and other design parameters.

This research aims to provide a deeper understanding of the stress distribution in non-centrally located nozzles, ultimately contributing to more reliable and optimized pressure vessel designs.

Jaroslav Mackerle [9] conducted a comprehensive bibliographical review of the finite element method (FEM) applied to the analysis of pressure vessel structures and piping. He examined research published between 2001 and 2004, referencing 856 papers and conference proceedings. The review covered various aspects, including:

- **Analysis types:** Linear and nonlinear, static and dynamic
- **Focus areas:** Stress and deflection analysis, stability, thermal issues, fracture mechanics, contact problems, fluid-structure interaction

- **Specific applications:** Manufacturing of pipes and tubes, welded pipes and pressure vessel components
- **Software and development:** Special finite elements for pressure vessels and pipes, finite element software

Mackerle's review highlighted the prevalence of topics like linear and nonlinear analysis, static and dynamic analysis, and fracture mechanics within the field of pressure vessel and piping research.

J. P. Balicevic, D. Kozak, and D. Karlievic [10] presented a study on the analytical and numerical solutions of internal forces in cylindrical pressure vessels with semielliptical heads. In this paper, a solution for internal forces and displacements in thin-walled cylindrical pressure vessels with ellipsoidal heads, using the general theory of thin-walled shell resolution, was proposed. The distribution of forces and displacements in the thin-walled shell was provided in mathematical form.

Finite element analysis of the cylindrical vessel with semielliptical heads was conducted using the ANSYS 10 code to validate the analytical solution. The ellipsoidal head model was designed as an axisymmetric problem to mitigate the bending effect on the contact between the heads and cylinders. The authors concluded that the principal stresses calculated analytically closely matched the results obtained from finite element analysis, with a difference of less than 3%.

K. M. F. Hsieh, D. G. Moffat, and J. Mistry [11] conducted a study on nozzles in the knuckle region of a torispherical head. In this paper, limit load interaction plots for pressure versus nozzle axial force, in-plane moment, and out-of-plane moment, as well as for in-plane moment versus out-of-plane moment, were presented. Six models were included in the study, with varying nozzle offset locations relative to the vessel outer diameter.

Model 1 represented the axisymmetric case with the nozzle located in the center of the crown, while Model 3 offset the outermost weld location at the crown/knuckle junction. Finite element models were created using the PATRAN mesh generation program, and stress analysis was performed using the ABAQUS program.

The authors concluded that the nozzle has minimal influence on the limit pressure of the head, even when located in the knuckle region. Additionally, for external loads applied to the nozzle, increasing the offset resulted in higher limit loads.

L. B. S. Thakkar and S. A. Thakkar [12] conducted a case study focused on designing a pressure vessel using ASME codes and standards to ensure compliance with legal requirements. They emphasized that assessing the performance of a pressure vessel under pressure entails conducting a series of tests according to the relevant ASME standard.

In their future scope, they proposed the utilization of PVELITE software for designing pressure vessels. Additionally, they suggested conducting finite element analysis (FEA) to validate the designed procedure.

The authors concluded that designing a pressure vessel is more about selecting suitable components rather than designing each component individually. Components for pressure vessels are chosen based on available ASME standards, and manufacturers adhere to these standards during production. This approach relieves designers from the burden of designing individual components, significantly reducing development time and the need for multiple prototypes. Using standard parts also facilitates quicker replacement, resulting in lower overall costs.

M. Shaik Abdul Lathuef and K. Chandra Sekhar [13] explored potential unintended consequences associated with the governing thickness of the shell as per ASME standards. They proposed a scope for modifying code values by considering the minimum governing thickness of the pressure vessel shell to meet specific requirements. Additionally, they suggested relocating nozzle positions to minimize stresses in the shell.
In their study, the authors conducted analyses using ANSYS software with nozzles located at five different positions: the left end, middle, and right end of the shell, as well as at both sides of the dished end. Stress calculations were performed based on these configurations.
Their findings indicated that stress levels were minimized at the dished end with a hillside orientation. A lower factor of safety resulted in material economy, leading to thinner, more flexible, and cost-effective vessels. The stress evaluation in the vessel was conducted using the Zick analysis approach.

RESEARCH GAP:
From existing literature, it's evident that the conventional design of pressure vessels relies heavily on ASME codes. Typically, pressure vessels encounter mechanical loads, making primary stresses the primary concern for designers. However, in certain applications like LMR (Liquid Metal Reactors), thermal loading plays a significant role. Therefore, this thesis aims to address the design and finite element analysis (FEA) of pressure vessels under thermo-mechanical loading conditions.

SCOPE OF WORK:
The topic presents a challenging yet promising avenue for future research and development. Here are potential areas for further exploration:
Comparative Study of Pressure Vessels: Investigate pressure vessels with various head types and conduct comparative research to assess their structural performance under different conditions.
Comprehensive System Design: Expand the focus to include the design and analysis of other critical components of pressure vessel systems to ensure a holistic understanding of their behavior and functionality.
Coupled Field Analysis Techniques: Implement advanced coupled field analysis techniques to explore the interactions between different components of the pressure vessel. This approach can provide deeper insights into their coupled thermo-mechanical behavior, thereby enhancing overall system performance.

REFERENCES
1. M. Jadav Hyder, M Asif, “Optimization of Location and Size of Opening In A Pressure Vessel Cylinder Using ANSYS”. Engineering Failure Analysis .Pp 1-19, 2008.
2. Joship Kacmarcik, Nedelijko Vukojevic, “Comparison Of Design Method For Opening In Cylindrical Shells Under Internal Pressure Reinforced By Flush (Set-On) Nozzles”. 2011

3. V.N. Skopinsky and A.B. Smetankin, "Modeling and Stress Analysis of Nozzle Connections In Ellipsoidal Heads Of Pressure Vessels Under External Loading." Int. J. Of Applied Mechanics And Engineering, Vol.11, No.4, Pp.965-979, 2006
4. J. Fang, Q.H. Tang, Z.F.Sanga, "Comparative Study of Usefulness for Pad Reinforcement in Cylindrical Vessels under External Load on Nozzle". International Journal Of Pressure Vessel And Piping 86,Pp 273-279, 2009
5. Pravin Narale, P.S. Kachare , "Structural Analysis Of Nozzle Attachment On Pressure Vessel Design," International Journal Of Engineering Research And Application,Vol.2,Pp 1353-1358 , 2012
6. James J. Xu, Benedict C. Sun, Bernard Koplik, "Local Pressure Stress On Lateral Pipe-Nozzle With Various Angles Of Intersection," Nuclear Engineering And Design 199, Pp 335-340, 2000
7. Arman Ayobstress , "Analysis Of Torispherical Shell With Radial Nozzle", The Institution Of Engineers, Malaysia, Vol. 67, 2006
8. V.N. Skopinsky, "Stress In Ellipsoidal Pressure Vessel Heads With Noncentral Nozzle," Nuclear Engineering And Design 198, Pp 317-323, 2000
9. Jaroslav Mackerle , "Finite Element In The Analysis Of Pressure Vessels And Piping, An Addendum: A Bibliography(2001-2004)," International Journal Of Pressure Vessel And Piping 82, Pp 571-592, 2005
10. P. Balicevic, D.Kozak, D. Kralievic, "Analytical and Numerical Solution of Internal Forces by Cylindrical Pressure Vessel with Semi-Elliptical Heads". First Serbian Congress On Theoretical And Applied Mechanics Kopaonik, Pp 10-13, 2007
11. M. F. Hsieh, D.G. Moffat, J. Mistry, "Nozzle In The Knuckle Region Of Torispherical Head: Limit Load Interaction Under Combined Pressure And Piping Loads", International Journal Of Pressure Vessel And Piping 77,Pp 807-815, 2000
12. B.S. Thakkar, S.A. Thakkar, "Design Of Pressure Vessel Using Asme Code, Section VIII Division 1",International Journal Of Advanced Engineering Research And Studies ,Vol I , Pp 228-234,(2012)
13. Shaik Abdul Lathuef, Chandra Sekhar, "Design and Structural Analysis Of Pressure Vessel Due To Change Of Nozzle Location And Shell Thickness", International Journal of Advanced Engineering Research and Studies, Vol. I, Pp 218-221,2012
14. Binesh P Vyas, R M Tayade, "Design Of Vertical Pressure Vessel Using Pvelite Software," International Journal Of Engineering Research And Technology, Vol. 2, Issue 3.
15. Drazan Kozak, Ivan Samardzic, "Stress Analysis Of Cylindrical Vessel With Changeable Head Geometry," Science Bulletin, Serie C, Vol. XXIII