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

DESIGN AND STRESS ANALYSIS OF PRESSURE VESSEL BASED ON ANSYS

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 critical component in many industrial plants, requiring thorough stress analysis to ensure safety and reliability. These vessels are subjected to both primary stresses from internal pressure and secondary stresses due to thermal loading, the latter being significant in vessels containing hot fluids. For instance, liquid metal reactors (LMRs) experience 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 is carried out using ANSYS software to get a more thorough knowledge of thermo-mechanical stressesWhen the outcomes from ANSYS and conventional analytical techniques are compared, it becomes clear how useful commercial finite element analysis (FEA) tools are for designing pressure vessels. Although FEA is not widely adopted in the industry for such analyses, this project demonstrates its potential advantages, particularly in handling coupled field analysis involving thermal and mechanical loads.

Keywords: Pressure Vessel, FE Analysis

1.Introduction

Pressure vessels are integral components in various industries, particularly in the nuclear sector, where they store fluids under conditions of high pressure and temperature. The design of these vessels is crucial, especially for reactor pressure vessels in advanced systems like liquid metal cooled nuclear reactors (LMRs). LMRs, which, because of their superior power density and safety advantages, have been thoroughly studied for power generation. They employ liquid metal as the main coolant and were originally designed for nuclear submarines. This work introduces a design method for pressure vessels, which are utilized in LMRs and operate at high temperatures and low pressures. Traditional design practices, primarily governed by ASME codes, focus on mechanical loadings and primary stresses. However, in the case of LMRs, thermal loading also plays a significant role As such, this work combines the design and finite element (FE) analysis of pressure vessels under thermo-mechanical loads, providing a comprehensive methodology that combines traditional approaches with state-of-the-art analytical tools.

PRESSURE VESSEL DESIGN

Pressure vessel design can be done primarily in two ways: Design by Rule and Design by Analysis. Design by Rule includes calculating the fundamental shell thickness and thermomechanical stresses and ensuring that they remain below allowable limits.

This method relies on the guidelines provided in ASME Section III, which is particularly relevant for pressure vessels used in nuclear applications. On the other hand, Design by Analysis provides a more detailed examination of stress distributions and structural integrity under operational conditions. The initial design created using the Design by Rule process is validated using finite element (FE) analysis. This research provides a comprehensive evaluation of the vessel's performance under thermo-mechanical loads.

The project concludes by comparing the results from both methods, highlighting the strengths and potential improvements offered by FE analysis in the design process.

2.1 Examination of a nuclear reactor using liquid metal cooling (LMR)

Table 1 gives an estimate of the size of the LMR application that was chosen for the investigation. It is believed that the pressure vessel's head is semi-elliptical.

Table 1: Approximate LMR Pressure Vessel Dimensions

Inner Diameter	11836 mm
Length of the Vessel	12001 mm

2.1.1 Material

SA-387 type 22 Class 2 chromium-molybdenum alloy is primarily designed for fabricators of welded pressure vessels and boilers intended for use in high temperature service.

Table 2 lists every property of the material.

Table 2: Characteristics of SA387

Yield Strength	236.145 N/mm2
Coefficient of thermal Expansion	13.994x10-5 mm/mm/K
Allowable Stress	94.45 N/mm2
Modulus of Elasticity	175.8x103 N/mm2

2.1.2 Operating conditions

Typical operating conditions are given in the table 3.

Table 3: Pressure vessel operating condition

Operating Pressure	1N/mm^2
Factor of Safety	2.5
Environmental temperature	423.15K
Operting Temprature	775.3K

2.2 Methodical Estimates

Each specification that is obtained is converted to a SI unit. All calculations in this work will now be performed in SI units.

Inputs for both units are shown in Table 4.

Table 4: LMR pressure vessel input specifications

Parameters	**SI Units**
Factor of Safety (FOS)	2.5
Operating Pressure (P)	1N/mm^2
Operting Temprature (T)	775.3K
Environmental temperature (T)	423.15K
Coefficient of thermal Expansion (α)	13.994x10-5 mm/mm/K
Allowable Stress (S)	94.45 N/mm2
Yield Strength (σ yield)	236.145 N/mm2
Density	7700 kg/mm^3
Modulus of Elasticity (E)	175.8x103 N/mm2
Length of the Vessel	12001 mm
Inner Diameter (D)	11836 mm

Table 5 displays the values of the various parameters, which were computed using accepted formulas found in the literature.

Parameters	**Value**
Vessel support thickness	15.25 mm
Total weight of pressure vessel	240360.3 Kg
Axial bending stress (considering simple temperature profile)	20.70 MPa
Meridional stress (Stress in Head),	93.15MPa
Circumferential Stress in Shell	93.70 MPa
Minimum semi ellipsoidal head thickness	63.5 mm
Minimum Shell Thickness	63.5 mm

Table 5: Various parameters calculated

2.3 Analytical design validation

2.3.1 Shell Thickness Design:

Equation 2.1 provided by ASME regulations is used to compute the minimum shell thickness. [6]

$$t = \frac{PR}{SE - 0.6P} \qquad (2.1)$$

t = 63.05mm ≈ **63.5mm (rounded off)**

2.3.2 Semi-Ellipsoidal Head Thickness Design:

ASME code Eq. 2.2 is used to compute the minimum head thickness.

$$t = \frac{PR}{2SE - 0.2P} \qquad (2.2)$$

t =62.72 mm

For a number of reasons, it is advised to utilize the same thickness—63.5 mm—for the head and shell when the difference in thickness is negligible.

2.3.3 Cylindrical Shell Circumferential Stress:

The design is checked for safety and stresses are recalculated after the minimum thickness of the shell and head has been established. 63.5 mm is the safe thickness.

$$\sigma = \frac{PRm}{t} \qquad (2.3)$$

= 93.70 MPa < 94.45 MPa, the permissible limit

2.3.4 Calculating Stress in a Semi-Elliptical Head:

The stress in the skull and the shell could differ.

As a result, a second assessment of head tension is required. Head stress is calculated using equation 2.4.

$$\sigma = \frac{PR2}{2th} \qquad (2.4)$$

Meridional stress (Stress in Head),

σ = 93. 15 MPa < allowable limit 94.45 MPa

2.3.5 Calculating Thermal Stress

For Simple Temperature Profile [16]

$$\text{Szb (Z)} = \frac{\sigma\text{zb (Z)}}{\text{E}\alpha\Delta\text{T}}\ [8] \qquad (2.5)$$

$$\boldsymbol{\sigma}\mathbf{zb} = \mathbf{20.70\ Mpa}$$

2.3.6 Membrane stress

$$\sigma_{\text{m}} = \left[\frac{\text{Ri(ri+t+}\sqrt{\text{(RmT)}}\text{+Ri(T+Te+}\sqrt{\text{r}_{\text{m}}\text{t)}}}{\text{A}_{\text{s}}}\right]\text{psi[8](2.6)}$$

$$\sigma_{\text{m}} = 27.44\text{Mpa}$$

3. FE ANALYSIS ANSYS [9] [10]

There are two methods for performing pressure vessel analysis:

i. Analysis of a quarter section using the cyclic symmetry technique

ii. analysis using thorough modeling and analysis of the vessel

The Axi-Symmetric Method

The Axi-Symmetric Approach makes use of symmetry along a particular axis to simplify the model and shorten computing times.

This method can be applied whenever the geometry can be rotated around an axis; in ANSYS, the Y-axis is usually the center of rotation.

As seen in Figure 1, the model is built and generated in ANSYS using PLANE components. This thesis consistently represents the shell, head, and complete model using the axi-symmetric technique for structural, thermal, and coupled field analysis.

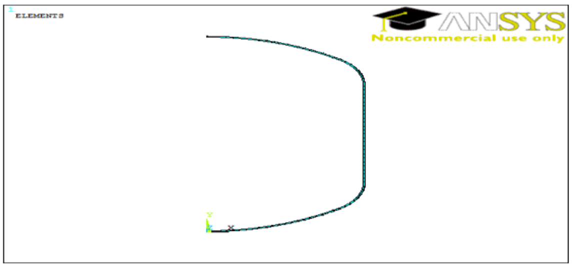


Figure 1: Complete Pressure Vessel Meshed Model. The head mesh model's zoomed view is displayed in the inbox.

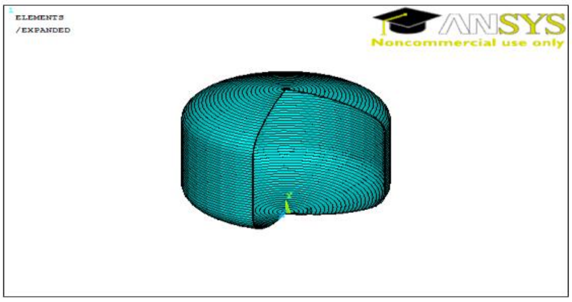


Figure 2: ¾ Axially symmetric

4. RESULTS AND DISCUSSION

In this chapter, the analytical results are presented and compared with those from the previous chapter.

The findings for the pressure vessel as a whole and the shell are provided separately.

4.1 Evaluation of Shell

For pure structural, thermal, and coupled field evaluations of the shell—that is, the portion of the pressure vessel that is not the heads—plane elements with axi-symmetry are employed.

Two-dimensional elements having translation degrees of freedom in the x and y axes are called plane elements.. These components might be quad or tria and usually have four nodes. The following are the specifics of the meshed model for the shell analysis:

1) Number of elements = 40
2) Number of nodes = 63
3) Element type = PLANE

4.1.1 Shell's structural analysis

The structural analysis use the PLANE42 element. Figure 3 displays the finite element analysis (FEA) model with applied loads and boundary conditions.

Modeling and meshing of the axi-symmetric shell cross-section are done using PLANE42 components.

The study is carried out to ascertain the stress distribution within the shell when pressure is applied to the inner surface.

Figure 4 depicts the stress distribution in the shell resulting from the pressure load, with the maximum stress observed at 1 N/mm^2.

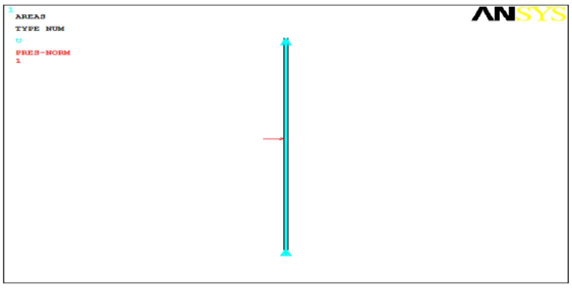
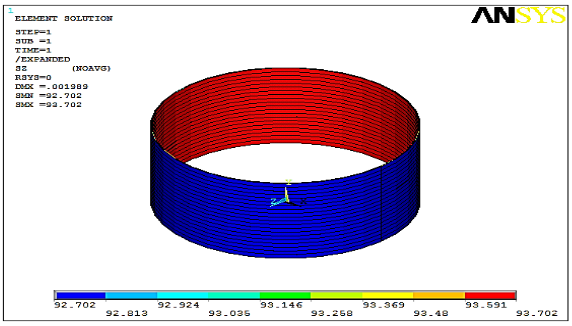


Figure 3: Loading and Limitations on Shell in Structural



```
ELEMENT SOLUTION
STEP=1
SUB =1
TIME=1
/EXPANDED
SZ          (NOAVG)
RSYS=0
DMX =.001989
SMN =92.702
SMX =93.702
```

Figure 4: Only the pressure load is causing stress in the shell (Full view)

Similar to the shell analysis, the pressure vessel analysis makes use of distinct axisymmetric elements for different kinds of analysis. In particular:

- Analysis of Structure: PLANE42 component
- Thermal Examination: PLANE55 component
- Analysis of Thermal Stress: PLANE42 Element
- Combined Field Study: PLANE13 component

The meshed model details utilized in the pressure vessel analysis are as follows:

- Number of elements = 200
- Number of nodes = 303

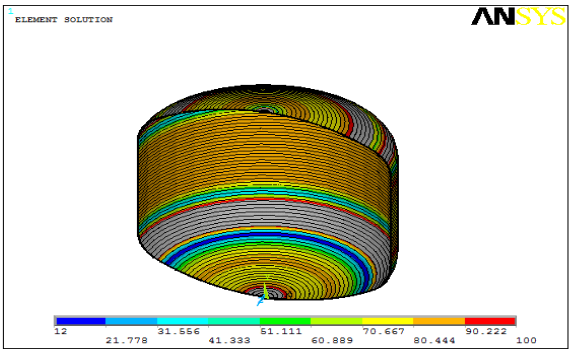
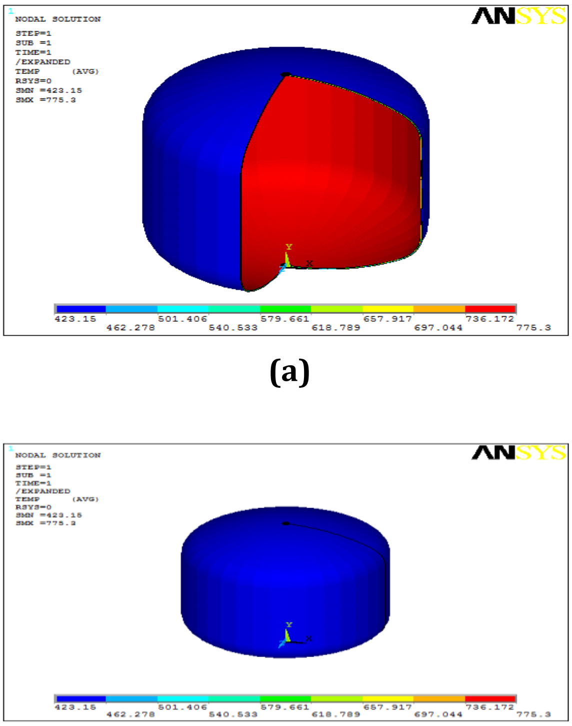


Figure 5: Structural Analysis of Pressure Vessel.



(a)

(b)

Figure 6: The Pressure Vessel's Temperature Distribution. (a) Partial View (b) Complete View

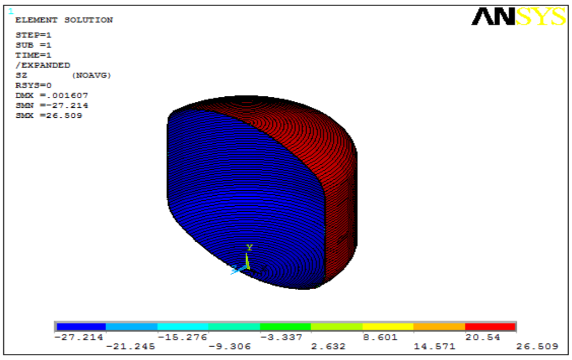


Figure 7: Analysis of Pressure Vessel Thermal Stress

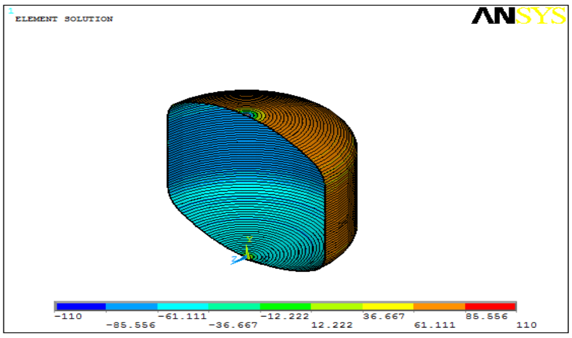


Figure 8: Coupled Pressure Vessel Field Analysis

5. A Comparison of Outcomes

Several analyses were performed on the pressure vessel's shell in order to compare the various analysis techniques. A summary of the results for the following analyses is shown in Table 6: Pure Thermal Analysis for Stress, Pure Structural Analysis, and Coupled Field Analysis and Pure Thermal Analysis for Temperature DistributionThe stress levels obtained from these investigations are found to be within the allowed limits specified by ASME regulations. A 15% discrepancy was found between the results of the finite element (FE) analysis and the analytical formulations; this discrepancy was attributed to factors such as approximations in the FE models and assumptions in the analytical formulations. Thus, a similar procedure can be used to analyze the pressure vessel overall, including the heads.

Table 7 displays the outcomes of the full pressure vessel modeling and analysis that was carried out in ANSYS utilizing axi-symmetric analysis. The stress values obtained show that the pressure vessel design is safe for use in LMR applications because they are far below the material's permissible limits. The study considered several situations and found stress levels of 100 MPa due to pure pressure loads and 26.5 MPa due to pure heat loads.Nevertheless, the combined impact led to somewhat larger stresses; the pressure vessel's coupled field analysis produced a stress of 110 MPa.

Table 6: Comparison of shell analysis results

Shell	Analytical (Mpa)	ANSYS (Mpa)	Percentage Error
Stress due to Pressure Loads	93.7	93.7	0%
Stress due to Thermal Loads	20.7	17.59	15%
Combined Stress due to Thermal & Pressure Loads	NA	103.07	NA

Table 7: Pressure Vessel Analysis Results

Pressure Vessel	ANSYS Stress Results (Mpa)
Combined Stress due to Thermal & Pressure Loads	110
Stress due to Thermal Loads	26.5
Stress due to Pressure Loads	100

Another finding from the entire pressure vessel study is that the junction of the shell and head is susceptible to bigger stresses than the individual stress levels in the shell and head. The abrupt change in geometry at the intersection is sometimes blamed for this phenomena. Because of this, it is essential to carefully analyze the stress concentration at this vital juncture when choosing the right head type for a specific pressure vessel design. In order to guarantee the pressure vessel's structural integrity and safety while it is in use, proper head selection is crucial.

6. RESULTS AND APPROACH FOR FUTURE RESEARCH

Conclusions:

The finite element analysis (FEA) tool is a vital tool in pressure vessel design as it provides valuable insights into the thermo-mechanical behavior of these structures. Several important results have been reached as a result of the current study:

- In order to endure the prescribed thermo-mechanical loads, the pressure vessel has been painstakingly developed and carefully studied, proving the adaptability and durability of FEA in engineering practice.
- The maximum stress caused just by pressure has been evaluated and compared with analysis values using ASME-established formulas. The maximum percentage error of 15% that was discovered highlights the precision and faithfulness of the FEA method.
- Notably, the safe operating conditions of the vessel have been confirmed using contemporary finite element analysis (FEA) methods, underscoring the role that FEA plays in ensuring both structural integrity and operational reliability.

Scope for Future Work:

The topic presents a challenging yet promising avenue for future research and development. The following list delineates potential areas for further exploration:

- Examining pressure vessels with various head kinds and carrying out comparative research to determine how well they function structurally under various circumstances
- Broadening the focus to encompass the design and study of other essential parts of pressure vessel systems in order to guarantee a thorough comprehension of their behavior and operation.
- Implementing coupled field analysis techniques in exploring the interactions between various accessories of the pressure vessel, offering deeper insights into their coupled thermo-mechanical behavior and enhancing overall system performance.

REFERENCES

1) http://en.wikipedia.org/wiki/Pressure_vessel
2) J.Phillip Ellen Berger, Robert Chuse, BryseE.Carson.Sr, “Pressure vessel” published by McGraw-Hill, 8th Edition
3)http://en.wikipedia.org/wiki/Liquid_metal_cooled_reactor
4) Frank Maslan, "Liquid-Metal Fuel Reactors: Part III", Brookhaven National Laboratory, Upton, New York, 1958
5) Rudolph J.Seavuzzo, “Pressure vessel”, published by CRC Press LLC, dated 2000
6) ASME Codes: Boiler Pressure Vessel Codes
7) Dennis Moss, “Pressure vessel” published by Gulf Professional Publishing, 3rd Edition
8) Ichiro Furuhasen, Nobuchika Kawasaki, Naoto Kasahara, “Evaluation charts of thermal stresses in cylindrical vessels induced by thermal stratification of contained fluid”, Volume 2, No. 4,dated 2008
9) NitinGokhale, Sanjay Deshpande, SanjeevBedekar, AnandThite, "Practical Finite Element Analysis", Finite to Infinite, India
10) www.ansys.com/