NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY
[IJIERT] ISSN: 2394-3696 Website: ijiert.org
VOLUME 10, ISSUE 10, Oct. -2023

EVALUATION OF COMBINED EFFECTS OF THERMAL AND MECHANICAL STRESSES ON THEPRESSURE VESSEL

Swati S. kumbar,
Prof. G.R. Deshpande
A.G Patil Institute of Technology Solapur. Maharashtra

Abstarct
One of the several plant components for which stress evaluations must be done is the pressure vessel. Primary stresses and secondary stresses are the two basic types of stresses that a pressure vessel endures. Pressure within the pressure vessel causes primary strains, while heat loading causes secondary stresses. An enormous amount of thermal stress occurs in a pressure vessel that handles hot fluid. Typically, thermo-mechanical loadings are present in liquid metal reactors (LMR). Induced stresses are determined analytically using pressure vessel ASME codes or theory. In this study, coupled field analysis for thermo-mechanical loading with ANSYS is used to compute induced stresses. The outcomes are next contrasted with analytical outcomes. The use of a commercial FEA tool instead of an analytical technique is demonstrated in this study. Thermal stress may be calculated using empirical connections provided by analytical solutions like ASME and Japanese Code. In the pressure vessel industry, using FEA tools is not particularly common. This type of loading necessitates coupled field analysis for FEA. In this study, coupled field analysis is carried out using ANSYS.

Keywords: Pressure Vessel, Thermo-Mechanical Stresses, ANSYS Coupled Field Analysis.

I INTRODUCTION

A pressure vessel is a closed container made to store gases or liquids at a pressure that is significantly higher than the surrounding atmosphere. These cylindrical containers are frequently used in a variety of industries as fluid storage containers. The fluid might be pressurized and operating at high temperatures.
The nuclear industry is one of pressure vessel's important applications [1] [2]. Reactor pressure vessels must be designed with special care. An advanced type of nuclear reactor called an LMR (liquid metal cooled reactor) uses liquid metal as its main coolant. Although extensively researched for power generation uses, liquid metal cooled reactors were initially developed for nuclear submarine use. Because the reactor doesn't need to be kept under pressure, they have safety advantages. and they enable a significantly higher power density compared to conventional coolants. The design methodology for pressure vessels used in LMR, which are subjected to low pressures and relatively high temperatures, is presented in this project. According to the literature, ASME codes are traditionally used in pressure vessel design. Additionally, since mechanical loadings are primarily applied to pressure vessels, primary stresses are constantly in the designers' minds. However, there aren't many applications, like LMR, where thermal loading is important. As a result, this work presents the design and FE analysis of a pressure vessel that was subject to thermomechanical loading.

II DESIGN OF PRESSURE VESSEL

There are two methods for designing vessels:
Designing using rules and analysis
Basic shell thickness, thermomechanical stresses, and keeping stresses below allowable values are calculated using design by rule. Since the chosen pressure vessel is used for nuclear applications, all designs are based

on ASME Section III [6]. The design created using the design by rule method is validated using FE analysis. Finally, the outcomes of these techniques are contrasted.

2.1 Analysis of Liquid metal cooled nuclear reactor (LMR) The application chosen for the study is LMR whose approximate dimensions are given in Table 1. Pressure vessel is considered with semi-ellipsoidal head.

Table 1: Approx. Dimensions of Pressure Vessel of LMR

Parameters	**SI units**	**Parameters**	**SI units**
Inner Diameter (D)	11836 mm	Co.Eff. of Thermal Exp. (**α**)	13.994x10 -5 mm/mm/K
Length of the vessel	12001 mm	Operating Temperature (T)	775.3 K
Modulus of Elasticity (E)	175.8x103 N/mm2	Environmental Temp. (T)	423.15 K
Density	7700 Kg/m3	Operating pressure (P)	1 N/mm2
Yield Strength (□Yield)	236.145 N/mm2	Factor of Safety (FOS)	2.5
Allowable Stress (S)		94.45 N/mm2	

2.1.1 Material

22th Grade SA-387 Specifically for use by fabricators in welded boilers and pressure vessels intended for use in elevated temperature service, Class 2 is a grade of chromium-molybdenum alloy. In table 2, all material characteristics are listed.

Table 2: Properties of SA-387

Modulus of elasticity	175.8x103 N/mm2
Yield strength	236.145 N/mm2
Allowable stress	94.45 N/mm2
Coefficient of thermal expansion	13.994x10-5 mm/mm/K

2.1.2 Operating conditions

Typical operating conditions are given in the table 3

Table 3: Operating conditions of pressure vessel

Operating Temperature	775.3 K
Environmental Temperature	423.15 K
Operating Pressure	1 N/mm2
Factor of Safety	2.5

2.2 Analytical Calculations

Every specification provided is converted to SI units. From now on, all calculations in this paper use SI units. Inputs for both units are shown in Table 4.

Table 4: Input specifications of pressure vessel used in LMR

Inner Diameter	11836 mm
Length of the Vessel	12001 mm

The different parameters have been calculated using the literature's standard formulas, and table 5 presents their results.

Table 5: Various parameters calculated

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

2.3 Validation of analytical design2.3.1 Design of Shell Thickness: Using Eq. 2.1 from the ASME codes, the minimum shell thickness is determined. [6]

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

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

2.3.2 Design of Semi-Ellipsoidal Head Thickness: Minimum head thickness is calculated using Eq. 2.2 given by ASME codes.

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

$$\mathbf{= 62.72\ mm}$$

When the thickness difference is very small, it is advised to use the same thickness for the head and shell, which is 63.5mm.

2.3.3 Circumferential Stress for Cylindrical Shell: Stresses are back calculated and the design is checked for safety after determining the minimum thickness for the shell and head. 63.5 mm is the assumed safe thickness.

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

= 93.70 MPa < allowable limit 94.45 MPa

2.3.4 Stress Calculations for Semi Ellipsoidal Head: Stress may not be equal in the head and the shell. Therefore, it is necessary to separately check for head stresses. Eq. 2.4 is used to calculate head stress**.**

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

Meridional stress (Stress in Head),

σ = 93. 15 MPa < allowable limit 94.45 MPa

2.3.5 Thermal Stress Calculation

For Simple Temperature Profile [16]

$$S_{zb}(z) = \sigma_{zb}(z)/(E\alpha\Delta T) \qquad (2.5)$$

σ_{zb}= 20.70 MPa

2.3.6 Membrane stress

$$\sigma_m = P\left[\frac{R_i(r_i+t+\sqrt{(R_m T)})+R_i\,(T+T_e+\sqrt{r_m t})}{A_s}\right] psi \text{ [8]}$$

σm =27.44MPa

2.3.7 Bending stress

$$\sigma_b = \frac{MC}{I}$$

σb= 20.58MPa

$\sigma_m + \sigma_b < S$ Therefore design is safe

III FE ANALYSIS USING ANSYS [9] [10]

There are two ways to analyze pressure vessels, which are described below:

Two methods of analysis were used: (i) a cyclic symmetry analysis on a quarter section, and (ii) analysis by drawing the entire vessel.

Axi-Symmetric Approach

The model is made easier to understand and takes less time to compute using an axi-symmetry approach. This method can be applied if the geometry is oriented around a specific axis. Axi-symmetry is used in ANSYS around the Y-axis. One must build the model depicted in Figure 1 and mesh it with ANSYS's PLANE elements. This thesis represents the shell, head, and entire model for structural, thermal, and coupled field analysis using an axisymmetry approach in every case.

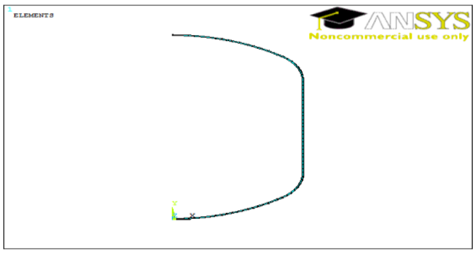


Figure 1:Meshed Model for Complete Pressure Vessel.

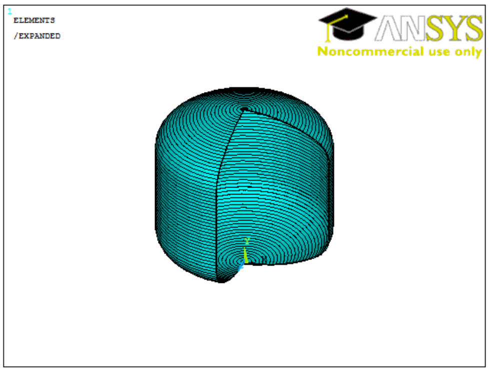


Figure 2: Axi-symmetric ¾ View

IV RESULTS AND DISCUSSION

The analysis results from the previous chapter are presented and contrasted in this chapter. Results for the shell and the entire pressure vessel are presented separately.

4.1 Analysis of Shell

The area of a pressure vessel without heads is called the shell. Plane elements with the axi-symmetry option are used for the pure structural, thermal, and coupled field analyses of the shell. A plane is a 2D plane element with x and y degrees of freedom in translation. Planes can be quad or triadic and have four nodes each.
The details of the mesh model made with plane elements are as follows: 1) Number of elements = 40 2) Number of nodes = 63 3)Element type = PLANE

4.1.1 Structural Analysis of Shell

The PLANE42 element is used for structural analysis. The FEA model with loads and boundary conditions is shown in Figure 3. The cross-section of an axisymmetric shell is modeled and meshes with PLANE42 elements. Inner surface pressure is applied, and shell stress is resolved. The stresses in the shell caused by pressure load are shown in Figure 4. 1 N/mm2 is the maximum stress.

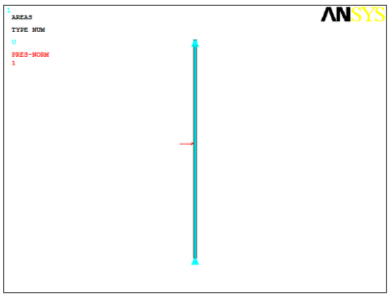


Figure 3: Loads and Boundary Conditions on Shell for Structural

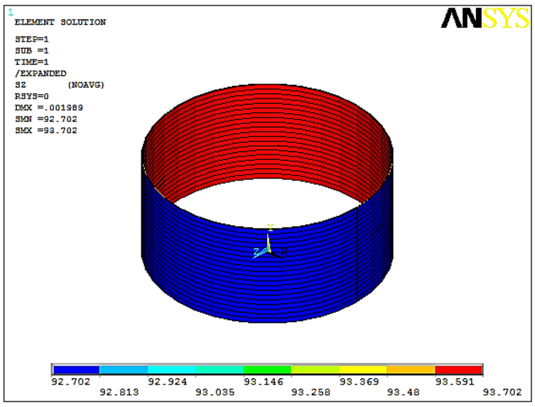


Figure 4: Stress in shell for due to pressure load only (Full view)

4.1.2 Thermal Analysis of Shell

The PLANE55 element is used for thermal analysis. Shell cross-section is modeled and meshed with PLANE55 elements for axi-symmetric analysis. The inner surface is heated to 775.3 K, while the outer surface is heated to 423.15 K. The temperature distribution in the shell caused by the difference in inner and outer temperatures is shown in Figure 5. The temperature distribution across thickness is depicted graphically in Figure 6.

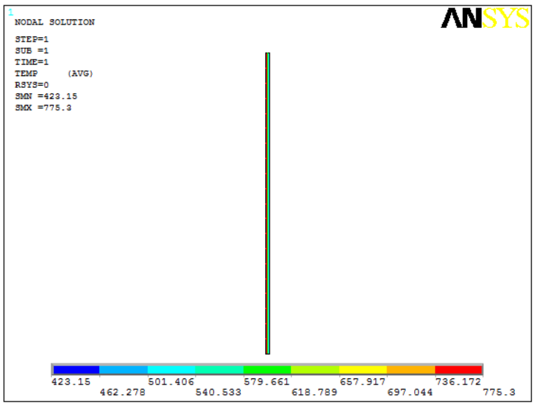


Figure 5.

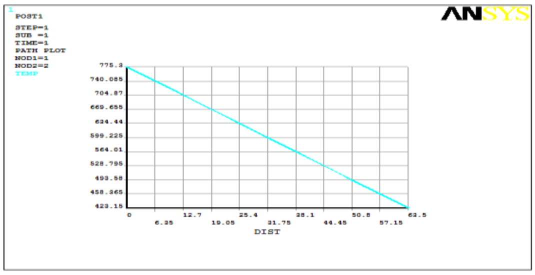


Figure 6: Temperature Distribution in Shell across Thickness.

4.1.3 Pure Thermal Stress Analysis of Shell

Using the PLANE42 element, a pure thermal stress analysis is performed. The cross-section of an axisymmetric shell is modeled and meshes with PLANE42 elements. In order to solve for stress in the shell caused by pure thermal loading without inner pressure, an inner temperature of 775.3 K is applied to the inner surface, while an outer temperature of 423.15 K is applied. The stresses in a shell caused by a pure thermal load are shown in Figure 7. 17.591MPa is the maximum stress.

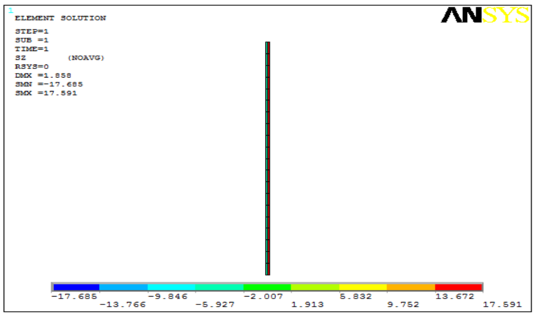


Figure 7: Stresses in Shell for due to Thermal Load Only

4.1.4 Coupled Field Analysis of Shell

Utilizing the PLANE13 element, structural-thermal coupled field analysis is performed. Coupled field analysis is a procedure that

Direct Coupling: Direct coupling typically entails a single analysis using a few field element types with all required DOFs. Direct coupling is utilized in the current study.

Figure 8 depicts the stresses in the shell caused by thermal and pressure loads. A 103.07 MPa maximum stress exists. Figure 9 depicts a typical full vessel used for analysis without heads**.**

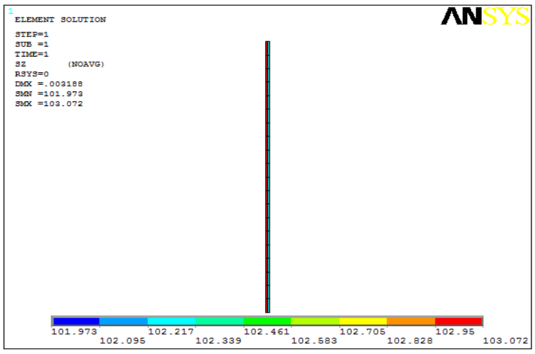


Figure 8: Stresses in Shell for Due Thermo-Mechanical Loads

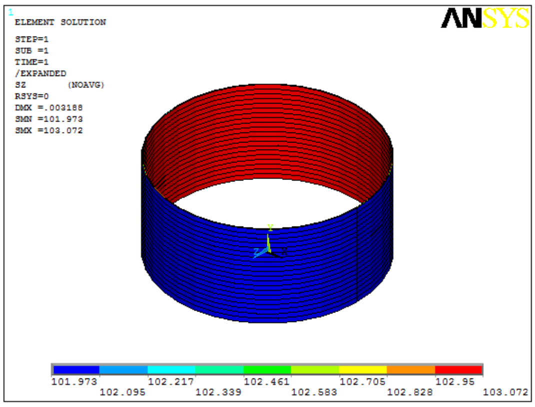


Figure 9: Stresses in Full Shell for Due Thermo-Mechanical Loads without Heads

4. 2 Analysis of Pressure Vessel

Similar to how shell analysis is conducted, pressure vessel analysis is also conducted. The following elements with the option for axi-symmetry are used for various pressure vessel analyses.

□□Structural Analysis: PLANE42

□□□□□□□□□□□□□Thermal Analysis: PLANE55

- Thermal Stress Anlysis: PLANE42
- Coupled Field Analysis: PLANE13

Following are the details of the meshed model using plane elements:

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

Various results are shown in Figure 10 through 16.

Figure 10: LBC's for Structural Analysis of Pressure Vessel

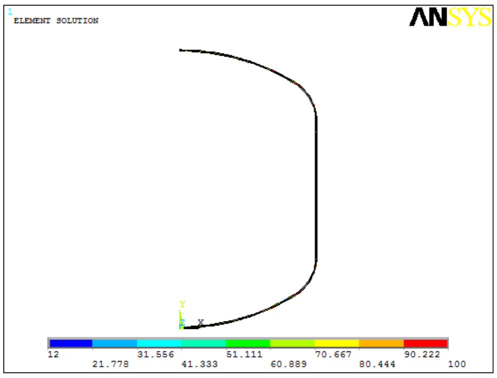


Figure 11: Stress due to pressure load on Pressure Vessel

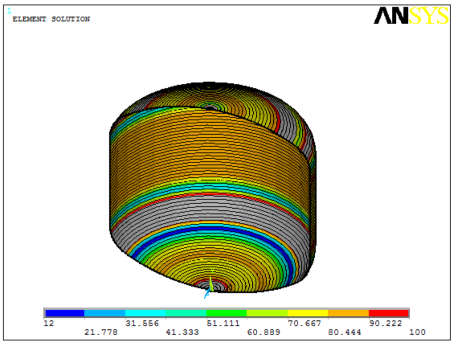


Figure 12: Structural Analysis of Pressure Vessel.

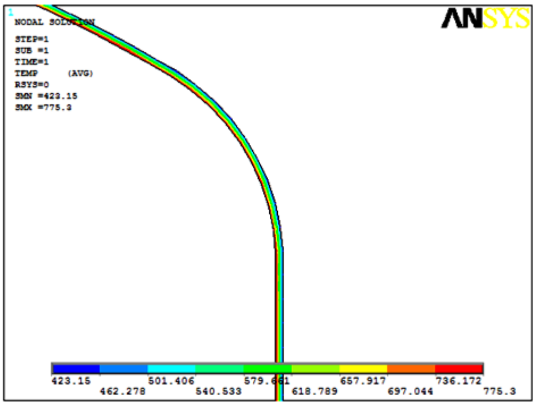
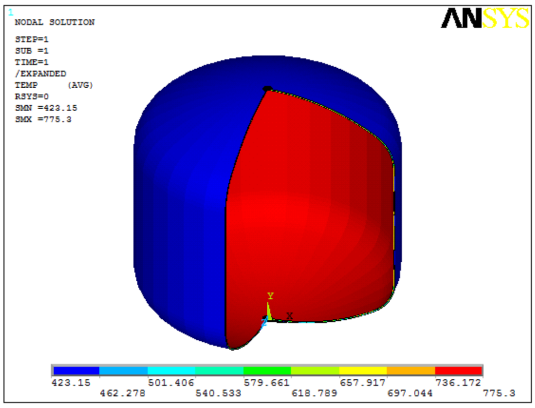
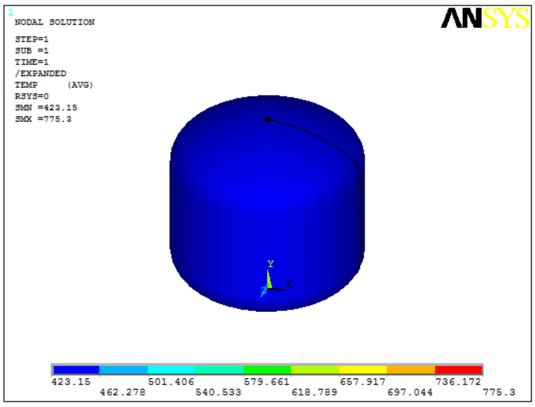


Figure 13: Temperature Distribution in Pressure Vessel

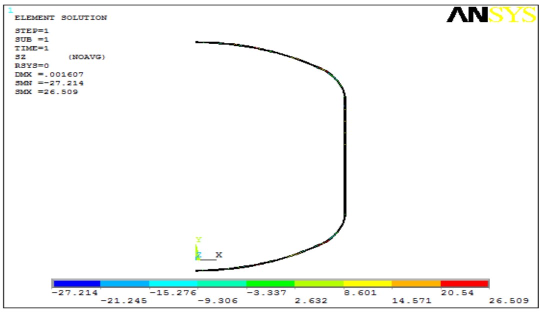


(a)



(b)

Figure 14: Temperature Distribution in Pressure Vessel. (a) 3/4th View (b) Full View



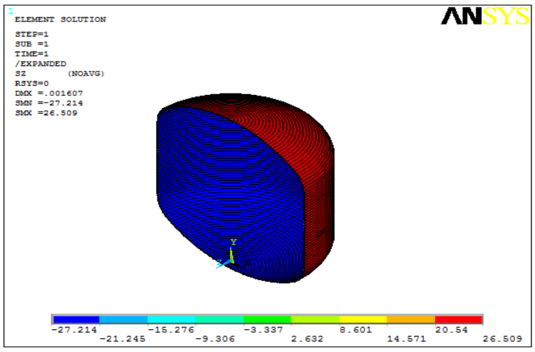


Figure 15: Thermal Stress Analysis of Pressure Vessel

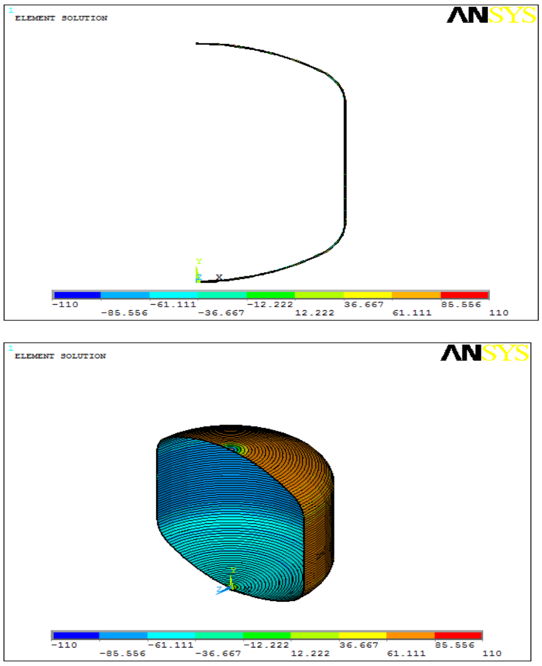


Figure 16: Coupled Field Analysis of Pressure Vessel

V COMPARISON OF RESULTS

The pressure vessel shell is subjected to a variety of analyses in the following to compare analysis methods. The outcomes are shown in Table 6.

- Pure Structural Analysis
- Pure Thermal Analysis for Temperature Distribution
- Pure Thermal Analysis for Stress
- Coupled Field Analysis

According to ASME guidelines, various stress results fall below the shell's permissible limits. When compared to analytical results, the results of the FE analysis are within a 15% error tolerance. There are many different causes of error in FE and analytical results, including approximations in FE formulations, assumptions in analytical formulation, element selection in FE analysis, etc. Therefore, the analysis of a pressure vessel along with its heads can be done using a similar procedure.

Results of various analyses are shown in table 7 after axi-symmetric analysis is used to model and analyze a complete pressure vessel in ANSYS. The pressure vessel is safe because the various stress values were less than the material's allowable limits, and this design can be used to produce pressure vessels for LMR. For the pressure vessel, the analysis was completed in ANSYS while taking various scenarios into account. Pure pressure load stresses are 100MPa, whereas pure thermal load stresses are 26.5MPa. The combined effect is slightly greater, though, and coupled field analysis results in 110MPa pressure vessel stresses.

Table 6: Comparison of results (of shell analysis)

Shell	Analytical (MPa)	ANSYS (MPa)	Percentage Error
Stresses due to Thermal Loads	20.70	17.59	15%
Stresses due to Pressure Loads	93.70	93.70	0%
Combined Stresses due to Thermal and Pressure Loads	NA	103.07	NA

Table 7: Results of Pressure Vessel Analysis

Pressure Vessel	ANSYS Stress Results (MPa)
Stresses due to Thermal Loads	26.5
Stresses due to Pressure Loads	100
Combined Stresses due to Thermal and Pressure Loads	110

The junction of the shell and head is subjected to high stresses in the complete pressure vessel analysis, whereas the pressure values for the shell and head separately are lower. Usually, a sudden change in geometry is the cause of this. Therefore, it's crucial to choose the right head type for the particular pressure vessel.

VI CONCLUSIONS AND SCOPE FOR FUTURE WORK

Conclusions:

The FEA tool can be used effectively in the design of pressure vessels. Understanding the thermo-mechanical behavior of pressure vessels typically aids the designer.

The following are the study's overall conclusions:

- The design and analysis of a pressure vessel is done for the specified thermomechanical loads.
- The maximum percentage error is 15% when the maximum stress induced by pressure alone in the shell is calculated using the ASME formula and compared with the analysis values.
- The vessel's safe operating conditions are confirmed within the framework of advanced FEA techniques.

Scope for Future Work:

The subject is difficult and offers a lot of room for future research. The list below outlines the potential for upcoming work:

- Comparing the outcomes of analyses of pressure vessels with various types of heads.
- Designing and analyzing additional pressure vessel parts
- Applying coupled field analysis to other pressure vessel accessories

REFERENCE

1.Frank Maslan, "Liquid-Metal Fuel Reactors: Part III", Brookhaven National Laboratory, Upton, New York, 1958.

2.Rudolph J.Seavuzzo, “Pressure vessel”, published by CRC Press LLC, dated 2000 .

3.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.

4.NitinGokhale, Sanjay Deshpande, SanjeevBedekar, AnandThite, "Practical Finite Element Analysis", Finite to Infinite, India.

5.J.Phillip Ellen Berger, Robert Chuse, BryseE.Carson.Sr, “Pressure vessel” published by McGraw-Hill, 8th Edition

6. ASME Codes: Boiler Pressure Vessel Codes

7.http://en.wikipedia.org/wiki/Liquid_metal_cooled_reactor

8.www.ansys.com/