International Journal Of Advance Research In Science And Engineering http://www.ijarse.com
IJARSE, Vol. No.2, Issue No.8, August 2013 ISSN-2319-8354(E)

COMPUTATION OF THERMO- MECHANICAL STRESSES IN PRESSURE VESSEL USING ANSYS

Basavraj R. Birajdar[1] **, Srinath E. Gudur**[2]

[1,2]Assistant Professor, Orchid College of Engineering & Technology,
Solapur University, Solapur, Maharashtra.(India)

ABSTRACT

The pressure vessel is one of a large number of plant components for which stress analyses must be performed. A pressure vessel experiences mainly two types of stresses, primary stresses and secondary stresses. Primary stresses are because of pressure inside pressure vessel and secondary stresses are because of thermal loading. Thermal loading is considerable in a pressure vessel which handles hot fluid. Typically liquid metal reactors (LMR) have thermo-mechanical loadings. Analytically, induced stresses are calculated using pressure vessel theory or ASME codes. In this paper induced stresses are calculated using ANSYS coupled field analysis for thermo-mechanical loading. The results are then compared with analytical results. This work demonstrates usage of commercial FEA tool over analytical approach. Analytical solutions such as ASME and Japanese Code give empirical relationships to calculate thermal stress. Use of FEA tools is not very popular in pressure vessel industries. FEA of this kind of loading requires coupled field analysis. In the work ANSYS is used to perform coupled field analysis.

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

I INTRODUCTION

A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. These cylindrical vessels are commonly used as containers to store fluids in various industries. The fluid may be at elevated temperatures and in a pressurized state.

One of the important applications of pressure vessel [1] [2] is nuclear industry. Special care has to be taken while designing reactor pressure vessels. A Liquid metal cooled nuclear reactor (LMR) [3] [4] is an advanced type of nuclear reactor where the primary coolant is a liquid metal. Liquid metal cooled reactors were first adapted for nuclear submarine use but have also been extensively studied for power generation applications. They have safety advantages because the reactor doesn't need to be kept under pressure, and they allow a much higher power density than traditional coolants. This project presents the design methodology for pressure vessel used in LMR which is subjected to low pressures and relatively high temperatures. From literature it appears that traditionally design of pressure vessel is carried out using ASME codes. Moreover, pressure vessels are mainly subjected to mechanical loadings hence; primary stresses are always focusing of the designers. But there is little application such as LMR where thermal loading is significant. Hence this work presents the design and FE analysis of pressure vessel subjected to thermo-mechanical loading

II DESIGN OF PRESSURE VESSEL

There are two ways in which vessels are designed:

i. Design by Rule ii. Design by Analysis

Design by rule is used to calculate basic shell thickness, thermo-mechanical stresses and keeping stresses below allowable values. All designs are based on the ASME Section III [6] as the selected pressure vessel belongs to nuclear applications. FE analysis is carried out to validate the design that is made by *design by rule method*. Finally the results obtained from these methods are compared.

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

Inner Diameter	11836 mm
Length of the Vessel	12001 mm

2.1.1 Material

SA-387 Grade 22 Class 2 is a grade of chromium-molybdenum alloy intended primarily for use of fabricators in welded boilers and pressure vessels which are designed for use in raised temperature service. All material properties are given in the table 2

Table 2: Properties of SA-387

Modulus of elasticity	175.8×10^3 N/mm^2
Yield strength	236.145 N/mm^2
Allowable stress	94.45 N/mm^2
Coefficient of thermal expansion	13.994×10^{-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/mm^2
Factor of Safety	2.5

2.2 Analytical Calculations

All input specifications are converted to SI unit. Henceforth, this paper follows SI units for all calculations. Table 4 shows inputs in both units.

Table 4: Input specifications of pressure vessel used in LMR

Parameters	SI units	Parameters	SI units
Inner Diameter (D)	11836 mm	Co.Eff. of Thermal Exp. (α)	13.994×10^{-5} mm/mm/K
Length of the vessel	12001 mm	Operating Temperature (T)	775.3 K
Modulus of Elasticity (E)	175.8×10^3 N/mm^2	Environmental Temp. (T)	423.15 K
Density	7700 Kg/m^3	Operating pressure (P)	1 N/mm^2
Yield Strength (σ_{Yield})	236.145 N/mm^2	Factor of Safety (FOS)	2.5
Allowable Stress (S)	94.45 N/mm^2		

The various parameters have been calculated using the standard formulae available in the literature and there values have been presented in the table 5.

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 design

2.3.1 Design of Shell Thickness: Minimum shell thickness is calculated using Eq. 2.1 given by ASME codes. [6]

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

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

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

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

= 62.72 *mm*

It is recommended to use same thickness for shell as well as head i.e.63.5mm due to several reasons when thickness difference is very small**.**

2.3.3 Circumferential Stress for Cylindrical Shell: After calculating minimum thickness for shell and head, stresses are back calculated and checked the design for safety. Safe thickness assumed is 63.5 mm.

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

= 93.70 MPa < allowable limit 94.45 MPa

2.3.4 Stress Calculations for Semi Ellipsoidal Head: Stress in head may not be same as in shell. Hence, check for stresses in head is separately required. Stress in head is calculated by Eq. 2.4

Meridional stress (Stress in Head), $\sigma = \frac{PR^2}{2th}$ (2.4)

σ = 93. 15 MPa < allowable limit 94.45 MPa

2.3.5 Thermal Stress Calculation

For Simple Temperature Profile [16]

$$S_{zb}(z) = \frac{\sigma_{zb}(z)}{E\alpha\Delta T} \quad [8] \qquad (2.5)$$

$$\sigma_{zb} = 20.70MPa$$

2.3.6 Membrane stress

$$\sigma_m = P\left[\frac{R_i\left(r_i+t+\sqrt{(R_m T)}\right)+R_i\,(T+T_e+\sqrt{r_m t})}{A_s}\right] psi \quad [8] \tag{2.6}$$

$\sigma_m = 27.44$MPa

2.3.7 Bending stress

$$\sigma_b = \frac{MC}{I} \tag{2.7}$$

$\sigma_b = 20.58$MPa

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

III FE ANALYSIS USING ANSYS [9] [10]

Pressure vessel analysis can be done in two different ways as mentioned below:

i. Analysis on a quarter section using cyclic symmetry approach

ii. Analysis by drawing the complete vessel

Axi-Symmetric Approach

Axi-symmetry approach simplifies the model and also reduces the computational time. This approach can be used if the geometry is revolved about a particular axis. In ANSYS axi-symmetry is used about Y-axis. One needs to create a model as shown in Figure 1 and mesh with PLANE elements of ANSYS. In all cases axi-symmetry approach is used in this thesis to represent shell, head and complete model for structural, thermal and coupled field analysis.

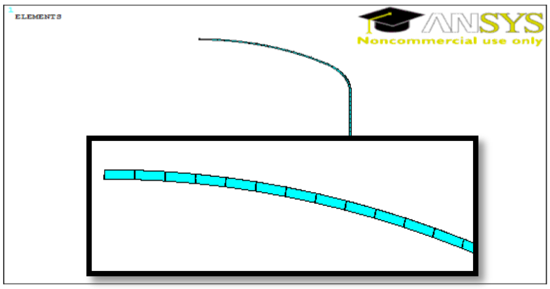


Figure 1:Meshed Model for Complete Pressure Vessel. Inbox shows zoomed view of the mesh model for head.

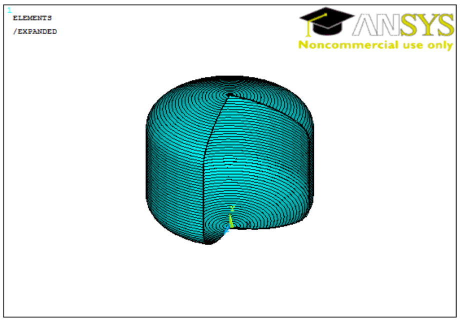


Figure 2: Axi-symmetric ¾ View

IV RESULTS AND DISCUSSION

In this chapter analysis results are presented and compared with analyitical results obtained in previous chapter. Results are presented seperately for shell and complete pressure vessel.

4.1 Analysis of Shell

Shell is the portion of pressure vessel without heads. Analysis of shell is carried out using plane elements with axi-symmetry option for pure structural, thermal and coupled field analysis. Plane is 2D plane element with translation x and y degrees of freedom. Plane has 4 nodes and it can be quad or tria.

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

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

4.1.1 Structural Analysis of Shell

Structural analysis is carried out using PLANE42 element. Figure 3 shows the FEA model with loads and boundary conditions. For axi-symmetric shell cross-section is model and meshed with PLANE42 elements. Pressure is applied inner surface and solved for stress in shell. Figure 4 shows the stresses in shell due to pressure load. Maximum stress is 1 N/mm^2.

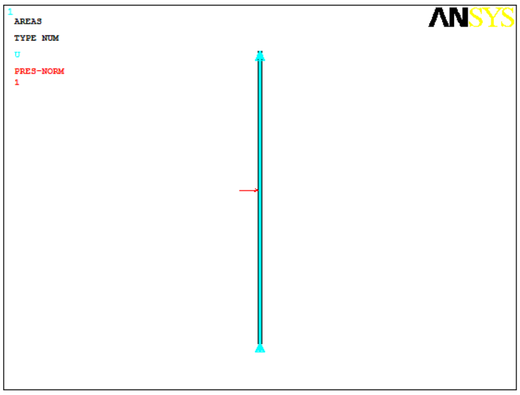


Figure 3: Loads and Boundary Conditions on Shell for Structural

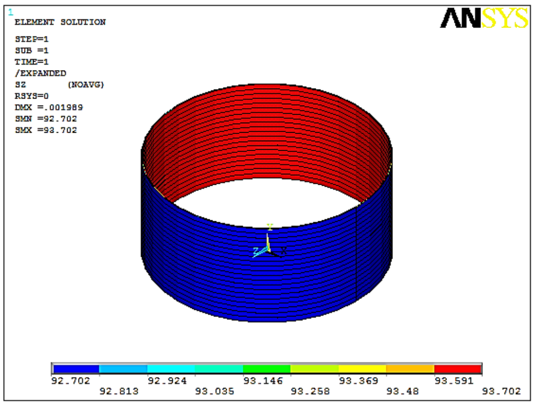


Figure 4: Stress in shell for due to pressure load only (Full view)

4.1.2 Thermal Analysis of Shell

Thermal analysis is carried out using PLANE55 element. For axi-symmetric analysis shell cross-section is modeled and meshed with PLANE55 elements. Inner temperature of 775.3 K is applied inner surface whereas 423.15 K is applied on outer surface. Figure 5 shows the temperature distribution in shell due to inner and outer temperature difference. Figure 5 shows the temperature distribution across thickness graphically.

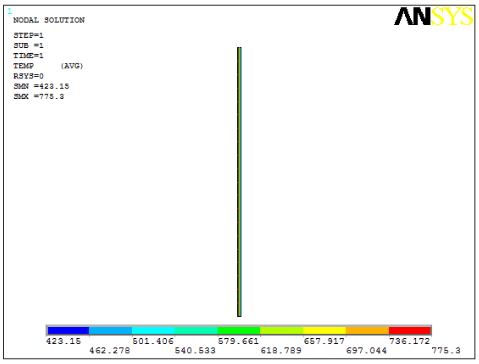


Figure 5: Temperature Distribution in

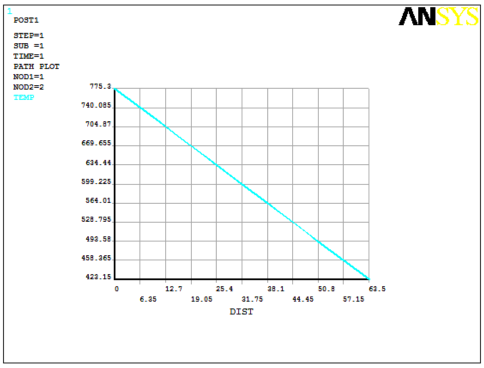


Figure 6: Temperature Distribution in Shell across Thickness

4.1.3 Pure Thermal Stress Analysis of Shell

Pure thermal stress analysis is carried out using PLANE42 element. For axi-symmetric shell cross-section is model and meshed with PLANE42 elements. Inner temperature of 775.3 K is applied inner surface whereas 423.15 K is applied on outer surface and solved for stress in shell due to pure thermal loading without inner pressure. Figure 7 shows the stresses in shell due to pure thermal load. Maximum stress is 17.591MPa.

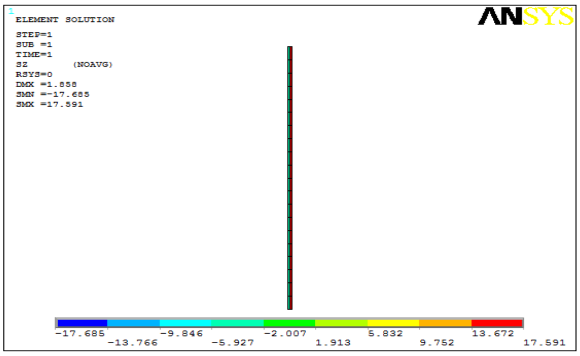


Figure 7: Stresses in Shell for due to Thermal Load Only

4.1.4 Coupled Field Analysis of Shell

Structural-Thermal coupled field analysis is carried out using PLANE13 element. Coupled field analysis can be carried out by

Direct Coupling: Direct coupling usually involves just one analysis that uses a couple field element types containing all necessary DOF's. In present study direct coupling is used.

Figure 8 shows the stresses in shell due to pressure and thermal loads. Maximum stress is 103.07 MPa. Figure 9 shows the typical full vessel without heads used for analysis.

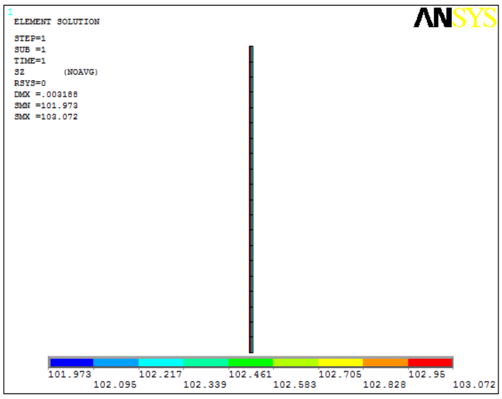


Figure 8: Stresses in Shell for Due Thermo-Mechanical Loads

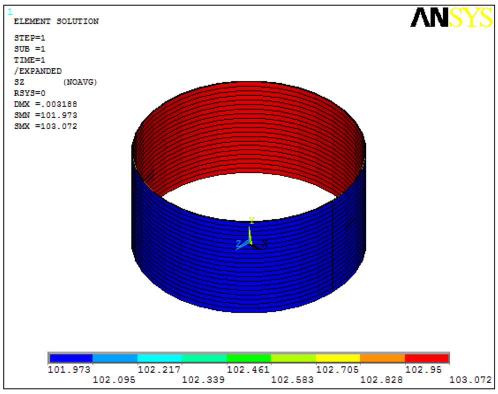


Figure 9: Stresses in Full Shell for Due Thermo-Mechanical Loads without Heads

4. 2 Analysis of Pressure Vessel

Analysis of pressure vessel is also carried out similar way the shell analysis is carried out. Following elements with axi-symmetry option are used for various analysis of pressure vessel.

- 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.

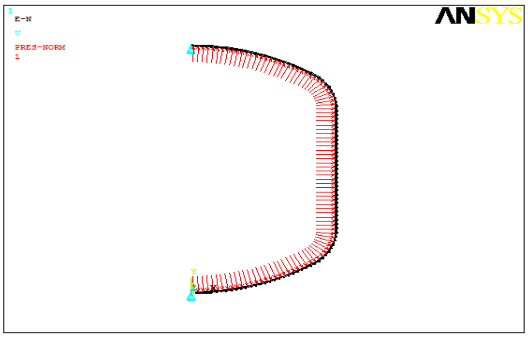


Figure 10: LBC's for Structural Analysis of Pressure Vessel

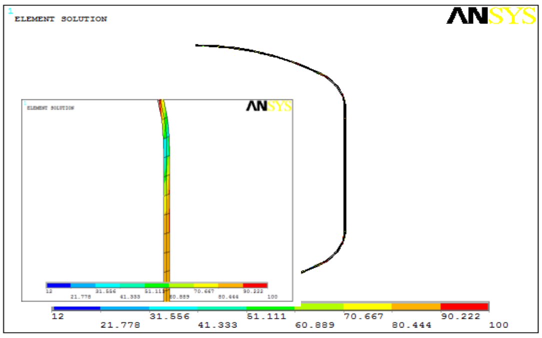
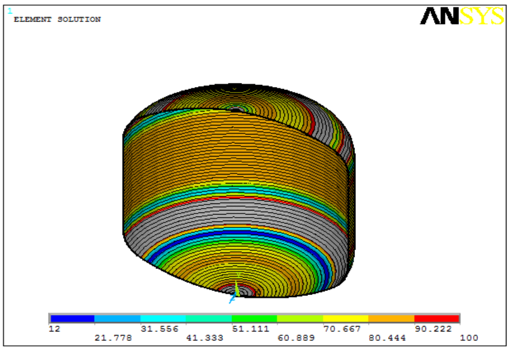


Figure 11: Stress due to pressure load on Pressure Vessel (Embedded Zoomed



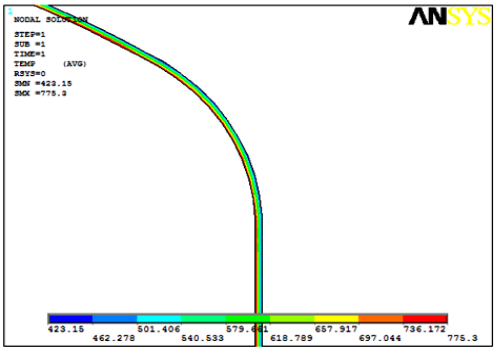
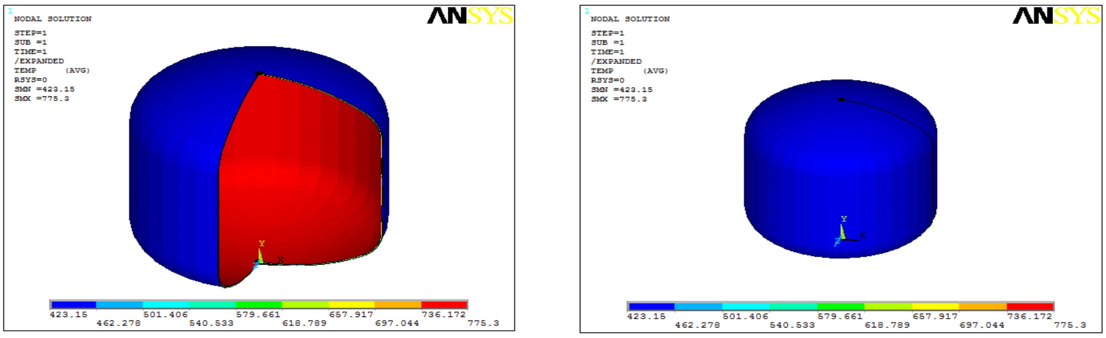


Figure 12: Structural Analysis of Pressure Vessel.

Figure 13: Temperature Distribution in Pressure Vessel (Embedded Zoomed



(a) *(b)*

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

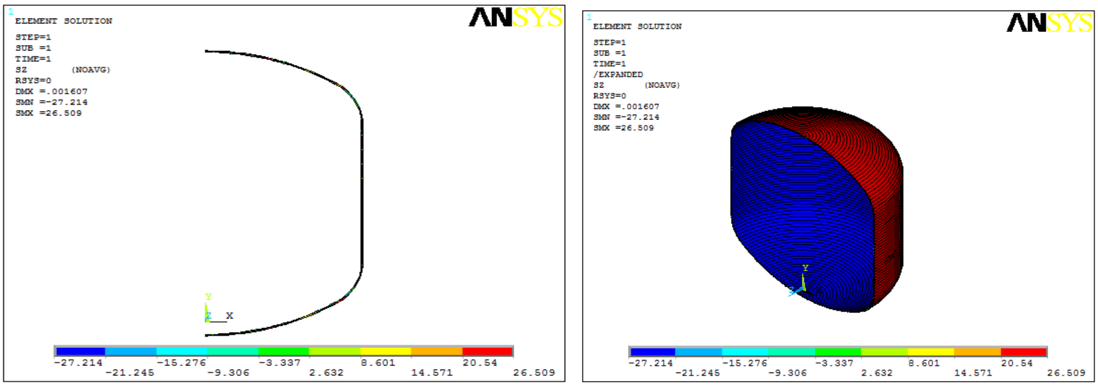


Figure 15: Thermal Stress Analysis of Pressure Vessel

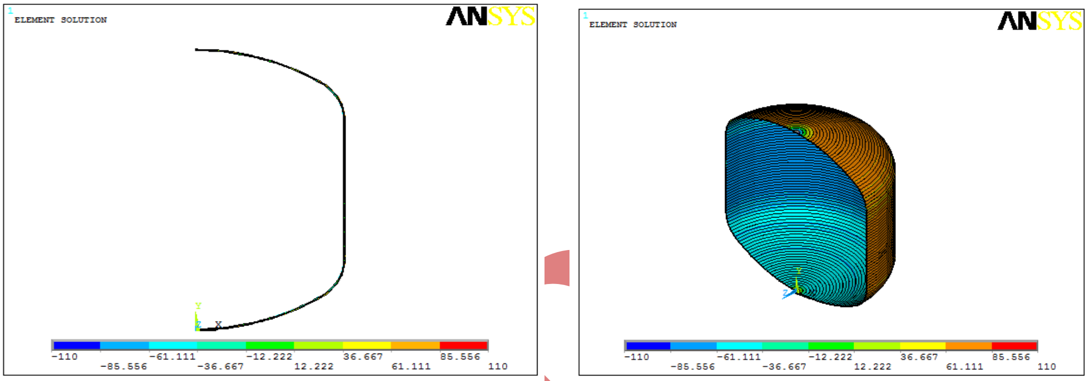


Figure 16: Coupled Field Analysis of Pressure Vessel

V COMPARISON OF RESULTS

Following various analyses are carried out on shell of pressure vessel to compare analysis procedure for pressure vessel analysis. Results are presented in Table 6.

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

Various stress results are below allowable limits of shell as per the guidelines of ASME. Results of FE analysis are within 15% error limit when compared to analytical results. Error in FE and analytical results occurs because of various reasons such as assumptions in analytical formulation, approximations in FE formulations, choice of element in FE analysis, etc. Hence, similar procedure can be used for analysis of pressure vessel along with heads.

Complete pressure vessel is modeled and analyzed in ANSYS using axi-symmetric analysis and results of various analyses are presented in table 7. Various stress values obtained are lesser than allowable limits of material and hence pressure vessel is safe and this design can be used to manufacture the pressure vessel to use in LMR. The analysis has been done in ANSYS for the pressure vessel taking different instances into consideration. Stresses due to pure pressure loads are 100MPawhere as stress due to pure thermal load is

26.5MPa. However, combined effect is little higher and coupled field analysis yields 110MPa stresses in pressure vessel.

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

Additional observation in complete pressure vessel analysis is that junction of shell and head is subjected to high stresses whereas shell and head individually indicates lower values of pressure. This is typically because of sudden change in geometry. Hence, it important to select appropriate head type for given pressure vessel.

VI CONCLUSIONS AND SCOPE FOR FUTURE WORK

Conclusions:

In design of pressure vessels FEA tool can be effectively used. Typically it helps the designer to understand thermo-mechanical behavior of pressure vessel. Overall conclusions based on present study are as below:

- Pressure vessel is designed and analyzed for the given thermo-mechanical loads.
- Maximum stress induced due to pressure alone in the shell is calculated using ASME formula and compared with the analysis values and the maximum percentage error is 15%.
- Safe operating conditions for the vessel are verified within framework of FEA advanced techniques.

Scope for Future Work:

The topic is challenging and involves lot of scope for future work. Following list outlines scope for future work:

- Analyzing pressure vessels with other types of heads and comparing results
- Designing and analyzing other components of pressure vessel
- Implementing coupled field analysis in other accessories of pressure vessel

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/