Static, Linear and Finite Element Analysis of Pressure Vessel

Prof. Vishal V. Saidpatil
Assistant Professor
Mechanical Engineering Department
NBNSSOE, Pune.
Vishal.saidpatil@sinhdad.edu

Prof. V.K.Kulloli
Assistant Professor
Mechanical Engineering Department
NBNSSOE, Pune.

***Abstract*— ‘Finite Element Method’ is a mathematical technique used to carry out the stress analysis. In this method the solid model of the component is subdivided into smaller elements. Constraints and loads are applied to the model at specified locations. Various properties are assigned to the A pressure vessel is a closed container designed to hold gases or liquids at a pressure different from the ambient pressure. The end caps fitted to the cylindrical body are called heads. The aim of this project is to carry out detailed design & analysis of Pressure vessel used in boiler for optimum thickness, temperature distribution and dynamic behavior using Finite element analysis software.model like material, thickness, etc. The model is then analyzed in FE solver. The results are plotted in the post processor. Paper involves design of a cylindrical pressure vessel to sustain 5 bar pressure and determine the wall thickness required for the vessel to limit the maximum shear stress. Geometrical and finite element model of Pressure vessel is created using CAD CAE tools. Geometrical model is created on Catia V5R19 and finite element modeling is done using Hypermesh. Ansys is used as a solver. Ansys APDL programming is used for number of simulation in linear static, modal and thermal analysis.**

1. INTRODUCTION

A. General Information

A pressure vessel is a closed container designed to hold gases or liquids at a pressure different from the ambient pressure. The end caps fitted to the cylindrical body are called heads.

Pressure vessels are used in a variety of applications. These include the industry and the private sector. They appear in these sectors respectively as industrial compressed air receivers and domestic hot water storage tanks, other examples of pressure vessels are: diving cylinder, recompression chamber, distillation towers, autoclaves and many other vessels in mining or oil refineries and petrochemical plants, nuclear reactor vessel, habitat of a space ship, habitat of a submarine, pneumatic reservoir, hydraulic reservoir under pressure, rail vehicle airbrake reservoir, road vehicle airbrake reservoir and storage vessels for liquefied gases such as ammonia, chlorine, propane, butane and LPG.

In the industrial sector, pressure vessels are designed to operate safely at a specific pressure and temperature technically referred to as the "Design Pressure" and "Design Temperature".

A vessel that is inadequately designed to handle a high pressure constitutes a very significant safety hazard. Because of that, the design and certification of pressure vessels is governed by design codes such as the ASME Boiler and Pressure Vessel Code in North America, the Pressure Equipment Directive of the EU (PED), Japanese Industrial Standard (JIS), CSA B51 in Canada, AS1210 in Australia and other international standards like Lloyd's, Germanischer Lloyd, Det Norske Veritas, Stoomwezen etc.

Pressure vessels can theoretically be almost any shape, but shapes made of sections of spheres, cylinders and cones are usually employed. More complicated shapes have historically been much harder to analyze for safe operation and are usually far harder to construct. Theoretically a sphere would be the optimal shape of a pressure vessel. Unfortunately the sphere shape is difficult to manufacture, therefore more expensive, so most of the pressure vessels are cylindrical shape with 2:1 semi elliptical heads or end caps on each end. Smaller pressure vessels are arranged from a pipe and two covers. Disadvantage of these vessels is the fact that larger diameters make them relatively more expensive, so that for example the most economic shape of a 1,000 liters (35 cu ft), 250 bars (3,600 psi) pressure vessel might be a diameter of 914.4 millimeters (36 in) and a length of 1,701.8 millimeters (67 in) including the 2:1 semi elliptical domed end caps.

Many pressure vessels are made of steel. To manufacture a spherical pressure vessel, forged parts would have to be welded together. Some mechanical properties of steel are increased by forging, but welding can sometimes reduce these desirable properties. In case of welding, in order to make the pressure vessel meet international safety standards, carefully selected steel with a high impact resistance & corrosion resistant material should also be used.

Two types of analysis are commonly applied to pressure vessels. The most Common method is based on a simple mechanics approach and is applicable to “thin wall” Pressure vessels which by definition have a ratio of inner radius, r, to wall thickness, t, of $r/t \geq 10$. The second method is based on elasticity solution and is always applicable regardless the r/t ratio and can be referred to as the solution for “thick wall” pressure vessels. Both types of analysis are discussed here, although for most engineering applications, the thin wall pressure vessel can be used.

II. LITERATURE REVIEW

A. History

The design of pressure vessels is an important and practical topic which has been explored for decades. Even though optimization techniques have been extensively applied to design structures in general, few pieces of work can be found which are directly related to optimal pressure vessel design. These few references are mainly related to the design optimization of homogeneous and composite pressure vessels.

L. P. Zick (1971) studied behavior of stresses in large horizontal cylindrical pressure vessels on two saddle supports. Zick indicates the approximate stresses that exist in cylindrical vessels at various locations and develop formulae to cover various conditions and charts.

B. Review of Papers

R. Carbonari, P Munoz-Rojas (et al 2011) discuses work on shape optimization of axisymmetric pressure vessels considering an integrated approach in which the entire pressure vessel model is used in conjunction with a multi-objective function that aims to minimize the von-Mises mechanical stress from nozzle to head. Representative examples are examined and solutions obtained for the entire vessel considering temperature and pressure loading. A proper multi-objective function based on a logarithmic of a p-root of summation of p-exponent terms has been defined for minimizing the tank maximum von-Mises stress [1].

V.N. Skopinsky and A.B. Smetankin describes the structural model and stress analysis of nozzle connections in ellipsoidal heads subjected to external loadings. They used Timoshenko shell theory and the finite element method. The features of the structural model of ellipsoid-cylinder shell intersections, numerical procedure and SAIS special-purpose computer program were discussed. A parametric study of the effects of geometric parameters on the maximum effective stresses in the ellipsoid-cylinder intersections under loading was performed. The results of the stress analysis and parametric study of the nozzle connections are presented [2].

Drazan ,Pejo, Franjo and Darko (2010) considered influence of stresses resulting from weld misalignment in cylindrical shell circumferential weld joint on the shell integrity .The stresses estimated analytically by API recommended practice procedure and calculated numerically by using the finite element method . [3]

L.You, J.Z hang and X. You present an accurate method to carry out elastic analysis of thick-walled spherical pressure vessels subjected to internal pressure. They considered two kinds of pressure vessels: one consists of two homogeneous layers near the inner and outer surfaces of the vessel and one functionally graded layer in the middle; the other consists of the functionally graded material only. They found that proposed approach converges very quickly and has excellent accuracy [7].

R. Patil, Dr. Bimlesh Kumar (et al 2011) studied the fracture pattern of perforated aluminum sheets experimentally and numerically using finite elements models on two different length scales, to understand effect of small scale features such as voids or hard particles on local deformation which have significant influence on the failure mode of a material .They found that the fracture path, successfully predicted without introducing softening material models. The softening phenomenon is naturally taken into account by the formation of localized deformation bands [8].

Many works including analytical, experimental and numerical investigations have been devoted to the stress analysis of nozzle connections in pressure vessels subjected to different external loadings.

C. Comments

Review of literature gives idea about various types of analysis of Pressure vessels. Designing a pressure vessel using a handbook is troublesome and not interactive. In this Paper further improvement achieve using following steps,

- Design Pressure Vessel as per Problem statement
- Geometrical model of Pressure vessel is created using CATIA V5 R19.
- Optimization analysis of pressure vessel is carried out for optimum wall thickness.
- Carried out Linear Static Structural analysis.
- Validate design Model using Finite Element Model.
- Carried out Dynamic analysis to identify the natural frequencies of loading that can be cause catastrophic destruction and find out critical mode shapes.
- Carried out Thermal analysis.
- Compare results of FEA with ASME boiler and pressure vessel regulations.

III. PROBLEM STATEMENT AND OBJECTIVES

A. Problem Statement

A cylindrical pressure vessel, as shown in Figure 3.1, is to be used for a boiler. The vessel consists of a cylindrical portion with the two ends closed using hemispherical structure. A nozzle is welded on at the mid-point of the length of the vessel which is supported on two supports. The vessel is constructed using rolled steel plates.

The internal pressure in the boiler is expected to be 5 bar. In addition, the flange of the nozzle is subjected to forces and moments being transmitted to the vessel through connected piping. The magnitudes of these forces and moments are given in Table 1.

Table 3.1 Forces and moments acting on the flange of the nozzle

Fx(N)	Fy(N)	Fz(N)	Mx(Nm)	My(Nm)	Mz(Nm)
1500	1000	2000	650	600	500

Fig 3.1 A typical cylindrical pressure vessel

Using closed form solutions for the cylindrical part of the vessel, determine the wall thickness required for the vessel to limit the maximum shear stress in the vessel to half the yield strength of the material used. Create a simple finite element model to analyze the configuration used for the closed form solution and check the accuracy of the closed form solution against the simulated results. Discuss how closing the open ends of the cylindrical tube affects the stress distribution in the vessel and, hence, its structural integrity. Create a detailed finite element model of the vessel using appropriate element type, size and order. Provide suitable justifications for the choices made. If auxiliary analysis is used to arrive at some decisions, provide short details of the same. Describe how the loads and boundary conditions have been applied and provide the reasons for the approach used. Comment, with appropriate figures, on the key features of the results and reasons for deviation from the closed form solutions. Suggest some methods for improving the identified shortcomings in design.

B. Objectives

- To review the literature on ASME pressure vessel design regulations, pressure vessel operation, and pressure vessel materials.
- To prepare 3 D model of the pressure vessel.

- To carry out closed form structural FE analysis to find optimum thickness to confine with ASME standard
- To carry out structural FE analysis of complete pressure vessel based on thickness calculated.
- To compare results obtained through FEA with ASME regulations.

C. Methodology

- Literature review for ASME pressure vessel regulations, pressure vessel operation and material used for pressure vessel will be carried out by referring journals, books, manuals and related documents.
- Based on reviewed literature, study will be carried out for pressure vessel critical characteristics, bottlenecks & regulatory requirements.
- 3 dimensional CAD model will be created based on blue prints using commercial CAD tool CATIA V5.
- Structural FE analysis will be carried out to find the optimum pressure vessel wall thickness using commercial FEM software ANSYS
- To carried out structural closed form FE analysis of complete pressure vessel based on thickness obtained to findout critical areas.

IV. DESIGN APPROACH OF PRESSURE VESSEL

A. Factors Influencing Pressure Vessel Design

Regardless of the nature of application of the vessels, a number of factors usually must be considered in designing the unit. The most important consideration often is the selection of the type of vessel that performs the required services in the most satisfactory manner. In developing the design, a number of other criteria must be considered such as the properties of material used, the induced stresses, the elastic stability, and the aesthetic appearance of the unit. The cost of fabricated vessel is also important in relation to its service and useful life.

B. Design of Pressure Vessels to Code Specification

American, Indian, British, Japanese, German and many other codes are available for design of pressure vessels. However the internationally accepted for design of pressure vessel code is American Society of Mechanical Engineering (ASME).

Various codes governing the procedures for the design, fabrication, inspection, testing and operation of pressure vessels have been developed; partly as safety measure. These procedures furnish standards by which, any state can be assured of the safety of pressure vessels installed within its boundaries. The code used for unfired pressure vessels is Section VIII of the ASME boiler and pressure vessel code. It is usually necessary that the pressure vessel equipment be designed to a specific code in order to obtain insurance on the plant in which the vessel is to be used. Regardless of the method of design, pressure vessels within the limits of the ASME code specification are usually checked against these specifications.

C. Development and Scope of ASME Code

In 1911, American Society of Mechanical Engineering established a committee to formulate standard specifications for the construction of steam boilers and other pressure vessels. This committee reviewed the existing Massachusetts and Ohio rules and inducted an extensive survey among superintendents of inspection departments, Engineers, fabricators, and boiler operators. A number of preliminary reports were issued and revised. A final draft was prepared in 1914 and was approved as a code and copy righted in 1915. The introduction to the code stated that public hearings on the code should be held every two years. In 1918, a revised edition of the ASME code was issued. In 1924, the code was revised with the addition of a new section VIII, which represented a new code for unfired pressure vessels.

D. Selection of The Type of Vessel

The first step in the design of any vessel is the selection of the type best suited for the particular service in question. The primary factors influencing this choice are
i. Operating temperature and pressure.
ii. Function and location of the vessel.
iii. Nature of fluid.
iv. Necessary volume for storage or capacity for processing.

It is possible to indicate some generalities in the existing uses of the common types of vessels. For storage of fluids at atmospheric pressure, cylindrical tanks with flat bottoms and conical roofs are commonly used. Spheres or spheroids are employed for pressure storage where the volume required is large. For smaller volume under pressure, cylindrical tanks with formed heads are more economical.

E. Material Selection

The Material selection is done considering the following factors:

- Strength
- Corrosion Resistance
- Resistance to hydrogen attack
- Fracture toughness

- Manufacturing (Fabrication)

The materials selected for the boiler is SA516 Gr 70, for which the yield stress is 260MPa and minimum tensile stress is 480MPa. ASME, Maximum allowable stress. According to given condition, to limit the shear stress to half of Yield strength of material, the design calculations are devised using the maximum shear stress theory. According to this theory the maximum shear stress is half the algebraic difference of the maximum and minimum principal stress. The maximum principle stress is the hoop stress (σ_{t1}) and minimum principle stress is the longitudinal stress (σ_{t2}).

$$\tau = (\sigma_{t1}) - (\sigma_{t2}) / 2$$
$$\tau = \sigma y / 2 = (\sigma_{t1}) - (\sigma_{t2}) / 2$$
$$\sigma y = (\sigma_{t1}) - (\sigma_{t2})$$
$$\sigma y = 2(\sigma_{t2}) - (\sigma_{t2}) = (\sigma_{t2}) \text{ ....since, } (\sigma_{t1}) = 2(\sigma_{t2})$$
$$\sigma y = 260 \text{ MPa} = (\sigma t2)$$
$$(\sigma t1) = 520 \text{ MPa}$$

The Hoop stress in a vessel is given by

$$(\sigma_{t1}) = p.\ d / 2.\ T$$
OR
$$T = p.\ d / 2.\ (\sigma_{t1})$$
$$T = (0.5 \times 1500) / (2 \times 520)$$
$$T = 0.7\text{mm}$$

The corrosion and weld efficiency, 1.5mm and 0.95mm respectively are added to above thickness, hence the thickness becomes, T = 3.17mm .The above value is rounded to 3mm, as the standards sheet size is available in 3mm.Hoop stresses are calculated for 3 mm thickness.

Hoop stress (σ_{hoop}),

$$\sigma_{hoop} = P.r/T$$
$$= 0.5*750/3$$
$$= 125\ (\text{N/mm}^2)$$

V. FINITE ELEMENT MODELLING

Modeling is the process of producing a model; a model is a representation of the construction and working of some system of interest. A model is similar to but simpler than the system it represents. One purpose of a model is to enable the analyst to predict the effect of changes to the system. On the one hand, a model should be a close approximation to the real system and incorporate most of its silent features. On the other hand, it should not be so complex that it is impossible to understand and experiment with it. A good model is a judicious tradeoff between realism and simplicity.

Geometrical model of Pressure vessel is as shown in figure 5.1

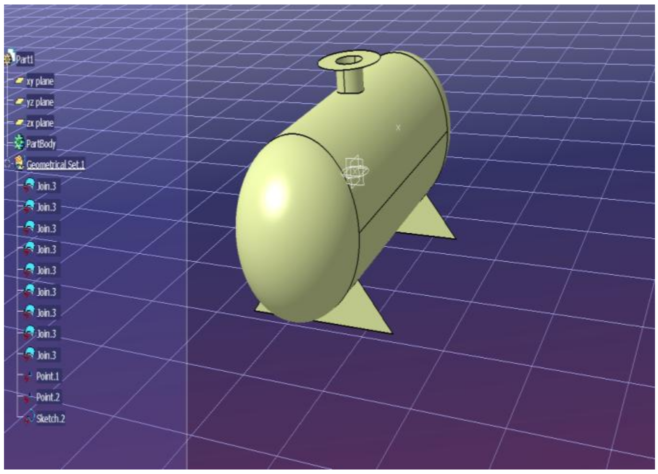

Fig 5.1 Geometrical model of pressure vessel

A. Pre-Processing

Finite element modeling is done using Hypermesh 9.0. Hypermesh is high performance finite element pre and post processor for major finite element solver helps to design condition in highly interactive and visual environment. Proper selection of element type, order and size plays important role in finite element analysis.

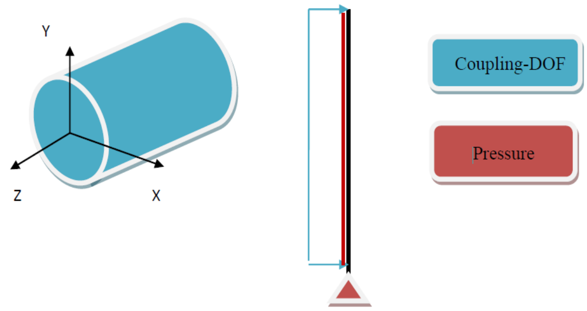

Fig 5.2 Axisymmetric FE Model

1). Element Type: For most of the pressure vessels the element selection is made from either axisymmetric solid elements, shell/plate elements or 3-D brick elements. However, 3D elements take much computation time than other two. As the vessel is to be modelled with the nozzle it cannot be treated as axisymmetric. Shell elements can represent curved surface. As the ratio of thickness of the vessel to radius (r) is less than 0.1, shell elements are used. The nozzle applies forces and moments in all three directions, i.e. in-plane and normal loads have to be considered while selecting the element type. Shell63 element is used as it as both bending and membrane capabilities.

2). Element order: As the geometry is not much complex, first order elements are used. Also the computation time taken is less compared to higher order elements.

3). Element size: As the stress concentration will be at the nozzle neck and cylinder intersection, it is modelled with denser mesh. Curvature at the nozzle area is studied with the chord deviation with different element size. The element size is selected to be 50 starting from the nozzle area. Mesh model of Pressure vessel is shown in figure 5.3

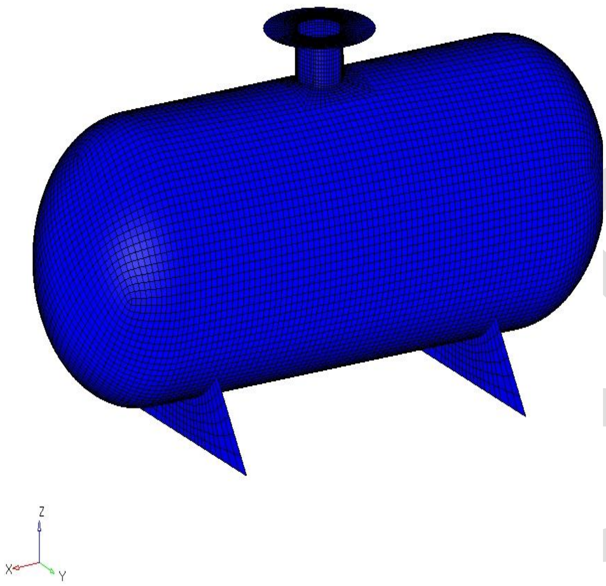

Fig 5.3 Finite Element Mesh Model

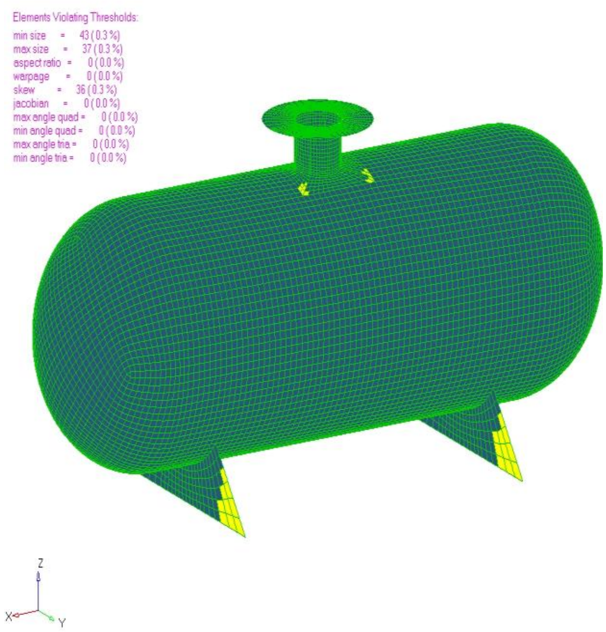

Fig 5.4 Quality Check of Finite Element Model

VI. STRUCTURAL ANALYSIS

Structural analysis consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes. Static analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

Design calculation done by the close formed solution needs to be validated with the Finite Element model. Hence, a simple finite element model is prepared by converting the pressure vessel into a 2D problem.

- Analysis type: Static Analysis
- Element used: 2-Node Finite Strain Axisymmetric Shell (SHELL208)
- The vessel being axisymmetric, and SHELL208 models thin axisymmetric shell structures, hence it is selected.
- The internal pressure generates translation in X direction.
- The element modelled is constrained at the centre node, and allowed to translate in X-direction.
- Loads: the internal pressure is taken as load and all over the element, in X-direction. The constrained are applied in Y-direction.
- Closed end consideration: The two hemispheres are taken for analysis as the pressure is applied throughout the vessel. For this only the shell is considered without nozzle and internal pressure of 0.5N/mm^2 is applied. The vessel is constrained at the bottom at two ends.
- The results for all the analysis are plotted using Von misses stress. As the material selected is ductile, the Von misses stress is used to check the yield strength. According to which for the design to be safe, Von Misses stress should be less than or equal to yield strength of material **(σv <= σy).**

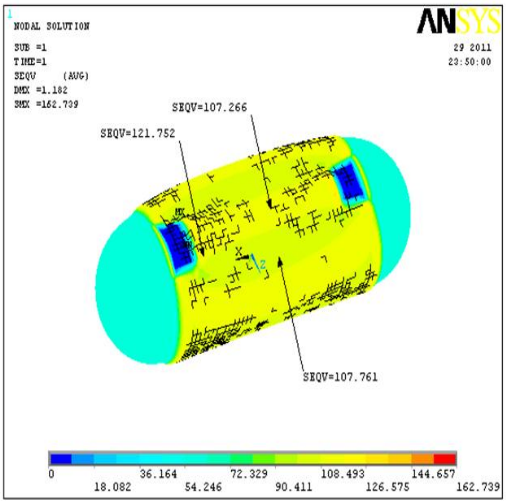

Fig 6.1 Stress in vessel with Closed Ends

Comments

- The stress values by using the 2d element matches exactly with the calculated stress.
- From the above figure it can be seen that the stress values lies between 110 to 127MPa as in yellow band. The stresses obtained from the analysis are tabulated below

Table 6.1 Comparison of Hoop stress value by calculations and analysis

	Closed Form	Analysis	Percent Deviation
Hoop stress	125MPa	127MPa	1.5%

The vessel consists of a cylinder portion, with two hemispherical ends and a nozzle with a flange is welded at the centre. The yield stress for the material is 260MPa.

- Assumptions: The vessel is analyzed with same thickness for cylinder and closed ends. The stress occurring at the supports are not considered.

- Inputs Loads: The nozzle is loaded with three forces and three moments. The vessel is subjected to internal pressure of 5 bars. The three forces and moments are applied on the flange inlet through mass node and rigid couplings.

- Boundary-conditions: Sub-modelling technique of Finite element is used which is based on the methodology of sub-regions, which contains singularities, geometry discontinuities or corrosion induced material degradation. The nozzle is taken as a sub-region with fine mesh while the rest model with coarse mesh. This method is known as the cut-boundary displacement method. In this the displacement calculated along the boundary of the coarse model is specified as boundary conditions for the sub-model.

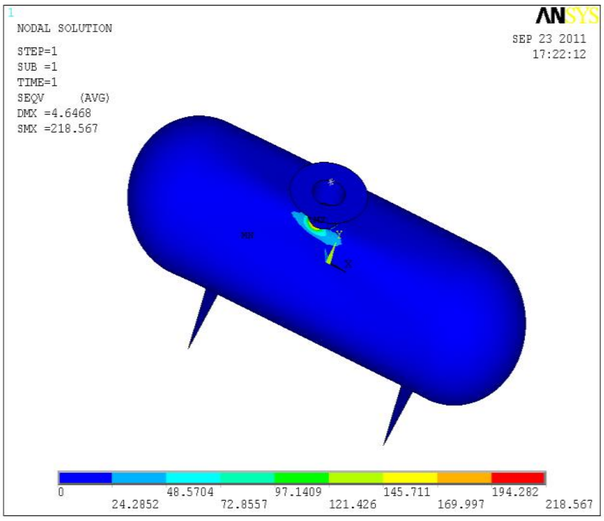

Fig 6.2 Von Misses Stress plot

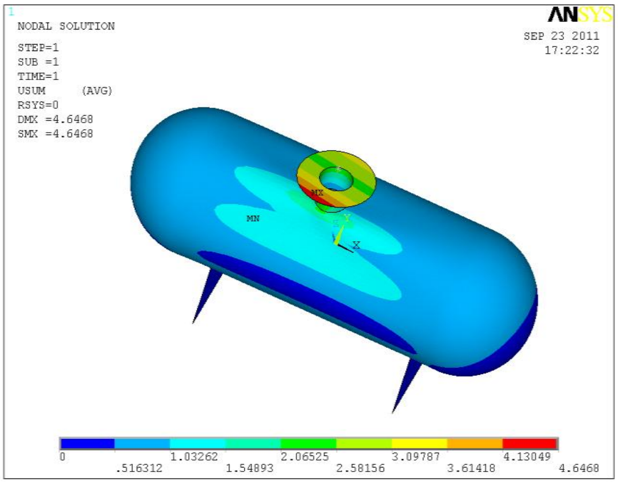

Fig 6.3 Nodal Displacement Plot

RESULTS

- Von Misses stress is calculated with thickness 3mm and 9mm to check the variation.
- It is observed that as the thickness increases the stress and displacement value decreases.
- For 3mm thickness the Von Misses stress is 215 MPa and that for 9mm is 69 MPa.
- Due to the application of the moments to the nozzle, the highest stress is indicated.

- **Using the Design load:** Analysis is done only with the loads acting on the nozzle. These are the primary stress, related to mechanical loading directly and satisfy force and moment equilibrium. These are well below the yield strength which will not fail due to mechanical loads.
- The overall displacement of the vessel is shown and is 4 mm. As the nozzle is loaded with forces and moments maximum deformation is observed at the flange.
- Also the displacement of 1 to 2 mm is seen at the neck area. This is because the thickness of the shell is much less than the nozzle to withstand the loads.

VII. STATUS OF THE PAPER

Design calculations shows thickness is 3 mm. Hoop stress corresponding to 3mm thickness is 125 MPa. Finite element model is created with element size 50 mm on Hypermesh 9.0, and linear static analysis is carried out to find deflection and stresses. Stress plot shows maximum value of von-mises stress is 127 MPa which is very near to 125 MPa. In such a way finite element model is validated. The closed form and analysis results are very close, with 1.5% deviation. The stress values are less than the yield stress of the material, which accounts for safe design.

REFERENCES

[1] Noraziah Wahi, Amran Ayob and Mohd kabashi Elbasheer, "Effect of Autofrettage on Allowable Pressure of Thick- Walled Cylinders", International Conference on Environment and Agriculture Engineering IPCBEE vol. 15 (2011), Singapore, Page no.14-17.

[2] Drazan Kozak, Franjo Matejicek, Darko Damjanovic, "Weld misalignment influence on structural integrity of Cylindrical Pressure Vessel", Structural integrity and life, Vol. 10, No 2 ,2010, Page no. 153-159.

[3] Alexey I. Borovkov, Dmitriy S. Mikhaluk, "Finite Element Stress Analysis and Multi-parameter Optimization of a High-pressure Vessel", Proc. 10th International ANSYS'2002 Conference Pittsburgh USA, 2002.

[4] R. Carbonari, P. Munoz-Rojas, E. Andrade, G. Paulino, K. Nishimoto, E. Silva, "Design of pressure vessels using shape optimization: An integrated approach", International Journal of Pressure Vessels and Piping, Volume 88, May 2011, Page no.198-212.

[5] R. Adibi-Asl, P. Livieri, "Analytical Approach in Autofrettaged Spherical Pressure Vessels Considering the Bauschinger Effect", Journal of Pressure Vessel Technology, Vol. 129, August 2007, Page no. 411-418.

[6] L. You, J. Zhang, X. You, "Elastic analysis of internally pressurized thick-walled spherical pressure vessels of functionally graded materials", International journal of Pressure Vessels and Piping, Volume 82, 2005, Page no. 347-354.

[7] R. Patil, Dr. Bimlesh Kumar, K. Waghulde, M. Patil, Dr. S. Mishra, "Failure Modes of Perforated Material under Finite Deformation for Pressure Vessel", International Journal of Advanced Engineering & Application, Jan-2011, Page no. 55-59.

[8] M. Y. H. Bangash, "Structures for Nuclear Facilities, Analysis, design and Construction", Published by Springer-Verlag Berlin Heidelberg 2011, Page no. 187-332.

[9] Dennis Moss, "Pressure Vessel Design Manual", Third edition, Elsevier, 2004.