Design and Optimization of Horizontal Pressure Vessel using FEM Analysis

Thesis is submitted in partial fulfilment of the requirements for the degree of

MASTER OF MECHANICAL ENGINEERING

By:

AROHI GUPTA

Examination Roll Number: M4MEC2201(O)

Registration Number: 133855 of 2015-16

Under the guidance of:

Prof. SAMAR CHANDRA MANDAL

&

Prof. NIPU MODAK

Department of Mechanical Engineering

Faculty of Engineering and Technology

Jadavpur University

Kolkata-700032

Design and Optimization of Horizontal Pressure Vessel using FEM Analysis

Thesis is submitted in partial fulfilment of the requirements for the degree of

MASTER OF MECHANICAL ENGINEERING

By:

AROHI GUPTA

Examination Roll Number: M4MEC2201(O)

Registration Number: 133855 of 2015-16

Under the guidance of:

Prof. SAMAR CHANDRA MANDAL

&

Prof. NIPU MODAK

Department of Mechanical Engineering

Faculty of Engineering and Technology

Jadavpur University

Kolkata-700032

Faculty of Engineering and Technology
Department of Mechanical Engineering
Jadavpur University
Kolkata-700032

DECLARATION OF ORIGINALITY AND COMPLIANCE OF ACADEMIC ETHICS

I hereby declare that this thesis contains literature survey and original research work by the undersigned candidate, as part of his Master of Mechanical Engineering (Machine Design) studies.

All information in this document have been obtained and presented accordance with academic rules and ethical conduct.

I also declare that as required by these rules and conduct, I have fully cited and referred all material and results that are not original to this work.

AROHI GUPTA

Examination Roll Number: M4MEC2201(O)

Registration Number: 133855 of 2015-16

Thesis Title: Design and Optimization of Horizontal Pressure Vessel using FEM Analysis

Signature: …………………….
(Arohi Gupta)

Dated: …………………………

Faculty of Engineering and Technology

Department of Mechanical Engineering

Jadavpur University

Kolkata-700032

CERTIFICATE OF APPROVAL

This forgoing thesis is hereby approved as a credible study of an engineering subject carried out and presented in a manner satisfactory to warrant its acceptance as a prerequisite to the degree for which it has been submitted, It is understood that by this approval the undersigned do not endorse or approve any statement made, opinion expressed or conclusion drawn therein but approve the thesis only for the purpose of which it has been submitted.

Committee on final examination for evaluation of the thesis

Signature: ………………

Date: …………………

Seal: ……………....

Signature: ………………

Date: ………………

Seal: ………………..

Signature: ………………

Date: ………………

Seal: …………...…

*Only in case, the thesis is approved.

Faculty of Engineering and Technology

Department of Mechanical Engineering

Jadavpur University

Kolkata-700032

CERTIFICATE OF SUPERVISION

We hereby recommend that the thesis presented by Mr. Arohi Gupta entitled **"Design and Optimization of Horizontal Pressure Vessel using FEM Analysis"** was under our supervision and is accepted in partial fulfilment of the degree of Master of Mechanical Engineering.

……………………………………

Prof. Samar Chandra Mandal
Thesis Advisor
Department of Mechanical Engineering
Jadavpur University
Kolkata

………..…………………………

Prof. Nipu Modak
Thesis Advisor
Department of Mechanical Engineering
Jadavpur University
Kolkata

Countersigned by:

…………………………….

(Head of the Department, Mechanical Engineering)

Date: ………………

Seal: …………..…

…………………………….

(Dean of Faculty of Engineering and Technology)

Date: ………………

Seal: …………..…

Acknowledgements

First of all, I would like to express my deep gratitude and admiration to my advisor and mentor Prof. Samar Chandra Mandal who is a constant source of guidance and inspiration to me. I am very grateful to you for the encouragement and support which you have provided throughout my time as your student. I have been extremely lucky to have a supervisor who cared so much about my work, and who responded to my questions and queries so promptly.

I want to extend my sincere appreciation and thanks to my co-guide, Prof. Nipu Modak who have been extremely helpful during the whole process of thesis review and have helped enrich my thesis with their valuable suggestions. I would like to mention a special thanks to Er. Pulak Kumar Roy from the CATIA, CREO, SolidWorks, ANSYS, AutoCAD Training Institute for providing me the basic supports required for the modelling and analysis. Their insight and expertise has greatly assisted this research. I would also like to thank my friends Vaibhav Gangwar, Shashikant Kumar, Prabhakar Chandesham, Mukul Anand and Shashi Kumar Prithwi for the encouragement they made in support of my research.

Most importantly I would like to dedicate this work to my parents and my wife who have been there for me always and are responsible for what I am today and without whom, the joy of completing the educational phase of my life would be incomplete. This accomplishment would not have been possible without them. Finally, I would like to express my gratitude to one and all, who directly or indirectly, have lent their hand in the support of my thesis work. Thank you

Abstract

The rupture of a pressure vessel may be quite dangerous, so the pressure vessels in industries are often constructed with a high safety factor. An extremely serious risk to life and property exists when a vessel is ineffectively or poorly built to manage high pressure. As a result, the ASME (American Society of Mechanical Engineers) Boiler and Pressure Vessel Code design codes are used to regulate the design and verification of pressure vessels. The goal of this thesis work is to reduce the overall stress and total deformation of a pressure vessel structure found in real life that is subject to the stress restrictions outlined in the ASME section VIII division-2 code.

Finding the most practical solution from among the available conventional solutions which accepts nearly any alternatives as long as they just meet the requirements of the problem is the process of optimization. The major goal of design optimization in pressure vessels is to save costs by lowering weight while maintaining adequate strength to prevent any design failure modes. This study examines axisymmetric pressure vessel size optimization from an integrated perspective.

The optimization process is carried out by integrating commercial finite element analysis software ANSYS modelling with CATIA software. In order to reduce the stress and overall deformation of the pressure vessel equipment, a model is combined with a single-objective function. The maximum linearized membrane and membrane plus bending loads are kept within the ASME code limitations while design factors like shell thickness and flange thickness are optimised.

Table of Contents

List of figures

List of tables

CHAPTER 1: INTRODUCTION

1.1 Short Information about pressure vessel

The most common engineering structures are pressure vessels, which are closed shells that store gases or liquids under internal pressure that can be found practically in any machine from a steam engine of the 18th century and a locomotive to a rocket motor and a spacecraft of the 21st century. Being fabricated mainly of steel, sometimes of titanium and aluminium alloys and, in special cases, of glass and plastics (bottles with carbonated water are also pressure vessels), the pressure vessels have gotten the greatest attention. In all of the nations where they are made and used, Design Guides, Codes, and Standards control their design, development, testing, and operation. There are two main reasons for that, the first one following from the possibility of such study and the second from the necessity of it. On one hand, pressure vessels, being usually shells of revolution, are described with relatively simple equations of shell theory, whose exact or rather accurate approximate analytical and numerical solutions can be readily found. On the other hand, such solutions are required for reliable prediction of the vessel response, because the uncontrolled failure of a vessel containing gas under high pressure results in explosion with often dire circumstances. As a result of operating experience. Accumulated over the ages, metal pressure vessels became the most reliable engineering structures, successfully used now in all the branches of modern industry.

As a result, it's critical to make sure the vessel is leak-proof and capable of withstanding the operational pressure and temperature. Pressure vessels range in size and shape from a simple beverage bottle to complex industrial pressure vessels. The pressure vessel's fundamental components are: i) the shell, ii) the head, iii) the nozzle and aperture, and iv) the support. The shell is normally cylindrical or spherical, whereas the head is flat, hemispherical, elliptical, or torispherical, among other shapes. The fluids are carried in and out of the vessel through the nozzles through the openings. Openings also enable for repair and maintenance activities while simultaneously weakening the shell owing to material withdrawal. As a result, openings act as stress raisers or stress concentration zones. Finally, the support is designed to keep the vessel afloat on a base.

1.2 Geometry of Pressure Vessel, Pressure Vessel Material and properties

The analysis in this thesis uses a standard nuclear pressure vessel. The dimensions chosen are typical of a pressure vessel used in nuclear power. Based on ASME standards VIII Division 1, the pressure vessel's geometry is selected. The length of the three-dimensional model was 5000 mm, and the radius of the shell was 1250 mm.

The following qualities of stainless steel, one of the materials used often for pressure vessels, were chosen as follows:

$Elastic\ modulus\ =\ 193\ GPa,$

$Yield\ strength\ =\ 207\ MPa,$

$Ultimate\ tensile\ strength\ =\ 493\ MPa,$

$Poisson's\ ratio\ =\ 0.31,$

$Permissible\ stress, S\ =\ 115\ MPa,$

$The\ operating\ pressure, P = 10\ MPa.$

The welding efficiency, E is assumed to be 1

Using ASME standards, the radius of the three head forms and the thickness of the shell are computed.

Shell thickness = $\frac{\mathrm{PR}}{SE-0.6P} = \frac{10\times1250}{115\times1-0.6\times10}$ = 115mm

Tori spherical head thickness = $\frac{0.885\mathrm{PL}}{SE-0.1P} = \frac{0.885\times10\times2500}{115\times1-0.1\times10}$ = 194mm

Ellipsoidal head thickness = $\frac{\mathrm{PD}}{2SE-0.2P} = \frac{10\times2500}{2\times115\times1-0.2\times10}$ = 110mm

Hemispherical head thickness = $\frac{\mathrm{PR}}{2SE-0.2P} = \frac{10\times1250}{2\times115\times1-0.2\times10}$ = 55mm

1.3 Pressure Vessel Supports

Depending on a pressure vessel's configuration, height to diameter ratio, placement convenience, operating temperature, and material, a variety of supports must be supplied. Four different types of supports are frequently employed.

1. Skirt support
2. Lug or Bracket support
3. Saddle support
4. Leg support

Heat exchangers and tall vertical columns are frequently supported on skirts. Medium sized process vessels are equipped with bracket or lug supports and have a height to diameter ratio of 2-3. Saddle supports are often offered for horizontal vessels. Leg supports are included with small vessels that have a capacity of 1-2 m^3.

1.3.1 Skirt support

Tall columns with skirt supports are available, such as those used in distillation and absorption. From the perspective of the designer, skirt support is desirable since it generates the least amount of local stresses as a result of mechanical loads operating at its intersection with the vessel.

In comparison to bracket supports, skirt supports are much easier to analyse. A skirt is a cylindrical shell with a diameter that is more than or equal to the vessel's outer diameter. It is welded to the vessel's bottom and sits on a bearing plate that is supported by a concrete base. The following loads are taken into account while designing skirt support:

1. Dead weight of the vessel.
2. Operating weight of the vessel.
3. Lateral loads caused by connected pipes' controlled thermal expansion.
4. Wind load acting on the vessel.
5. Seismic load

All of these loads must be taken into account when calculating the skirt thickness. Fig. 1.1 and Fig. 1.2 shows Lug support and skirt support respectively.

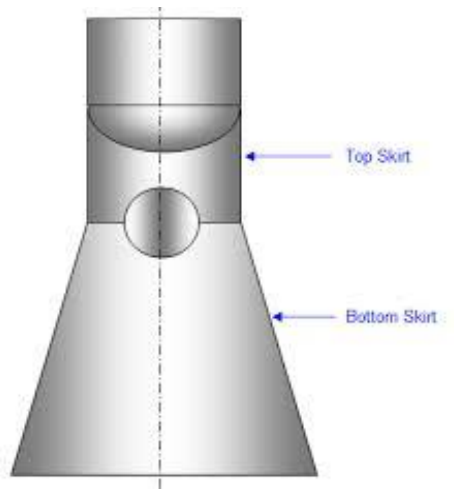


Fig. 1.1: Lug support

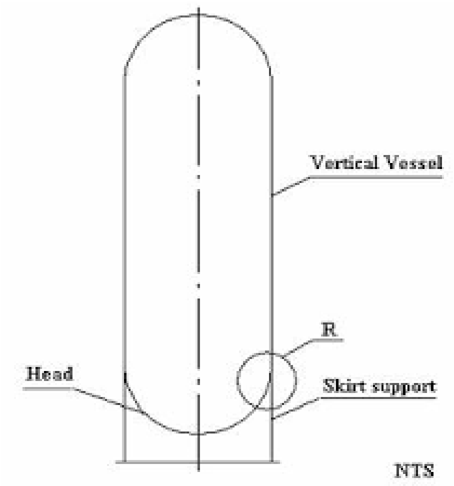


Fig. 1.2: Skirt support

1.3.2 Bracket or lug support

Process equipment frequently uses bracket supports. In general, bracket supports are supplied for vertical vessels with a height to diameter ratio of 2-3. These are made from plates and fastened to the vessel using the shortest amount of welding. On columns or support beams, brackets are supported. The image below shows a vessel with bracket support. The bracket supports provide a lot of benefits.

Less expensive.

It may be quickly and with a short weld length fastened to the vessel.

It is simple to level.

If a sliding arrangement is offered, it can absorb diametrical expansions.

Due to their capacity to absorb bending stresses caused by eccentric loads, thick wall vessels are the most appropriate.

There are also some drawbacks such as:

The vessel is eccentric with brackets. This causes the vessel wall to experience compressive, tensile, and shear stresses. Therefore, unless the vessel wall is strengthened with support plate, they are not appropriate for thin wall vessels. Fig. 1.3 shows Bracket or lug support which is used in vertical vessel prominently.

For vessel up to 0.8 m diameter two brackets are sufficient. Up to 3 m diameter, 4 brackets are used 6 brackets up to 5 m diameter and 8 brackets above 5 m diameter are used.

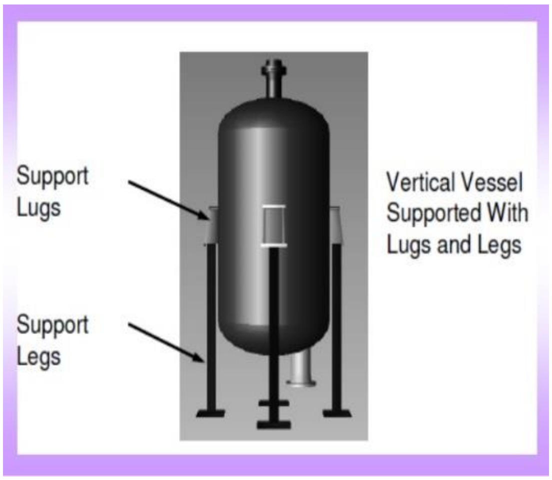


Fig. 1.3: Bracket or lug support

1.3.3 Saddle support

Saddle supports are typically supplied for horizontal vessels. Steel saddles sitting on top of concrete piers can be employed if the underside of the vessel is only to be situated a small distance above the ground line. Fig. 1.4 depicts horizontal vessels lying on a saddle support. The boat acts like a beam. Typically, there are two saddles. If necessary, it may be higher. If a support is needed in more than two places, ring-shaped supports are chosen. In selecting the location of saddle supports, advantage of stiffening effect offered by the head is taken into account.

Fig. 1.4: Saddle support

1.4 Design process and optimization

Optimization in general can be defined as the act of finding the best feasible solution with the most cost effective or highest achievable performance under the given constraints, by maximizing desired factors and minimizing undesired ones. The increased demand to cut down the production and manufacturing costs in the industries have encouraged engineers to use more robust decision making technique such as optimization. In any optimization problem, we seek values of the design parameters that minimize or maximize the objective while satisfying constraints. Over a past few decades, the optimization techniques have found its applications in a wide variety of industries such as automotive, aerospace, chemical, electrical and manufacturing industries. Because of the advancement in computer technology, the complexity of a problem being solved by optimization methods is no longer a concerned issue. Although this process can sometimes be very time consuming, depending on the size and nature of problem in hand. With the advent of computers, engineers can exploit and implement this procedure in practice. To fully realize the power of computational Design Optimization, it is important to implement optimization methods through pertinent computer-based mathematical tools.

CHAPTER 2: LITERATURE SURVEY

2.1 History

In the industrial sector, pressure vessels are used as storage tanks, diving cylinders, recompression chambers, distillation towers, autoclaves, nuclear reactor vessels, habitats for spaceships and submarines, pneumatic reservoirs, hydraulic reservoirs under pressure, rail vehicle airbrake reservoirs, road vehicle airbrake reservoirs, and storage tanks for liquefied gases, among many other vessels. A significant and useful subject that has been studied for decades is the design study of pressure vessels. Although optimization techniques have been widely used to design structures generally, there are very few works that specifically address optimization of pressure vessels by interfacing different software packages like CATIA and ANSYS. These few references include the design optimization of homogeneous as well as composite pressure vessels with different optimization methods.

2.2 Literature Review

Chattopadhyay [1] in his book mentioned that openings are used to transport fluids through nozzles into and out of the vessel. Aside from allowing for maintenance and repair work, openings can weaken the shell by removing material. Openings act as stress sources or areas where tension is concentrated. The support's final purpose is to help the vessel rest on a base. The support must be strong enough to support both the dead weight and any additional loads. Lug, skirt, and saddle are the many forms of supports.

The literature discussed by Hasan et.al. [2] has individually undertaken pressure vessel stress analyses for various situations, such as various head forms, supports, etc. In this section, a few of them are briefly summarised. To begin with, the pressure vessel's design was improved to reduce weight while maintaining strength and rigidity. Ant colony optimization technique was used to change the vessel's dimensions, including its thickness, length, and radius.

Y.K Sahu and S.Nagpal [3] done an optimization exercise performed in using finite element Analysis.

V.V Saidpatil and A.S. Thakare [4] performed optimisation technique using finite element package.

Widiharso et.al [5] also done similar work of optimisation using finite element package.

D.N. Bhinde and S. Rajanarasihma [6] worked in the stress analysis of the vessel, in order to determine which head shape is best, the effects of several heads including hemispherical, flat, torispherical, elliptical, and conical heads with the same cylinder thickness and volume were examined. The torispherical head was least strained, according to the findings of the finite element study.

Prasanna et.al. [7] found that the pressure vessel design was made to be as material-friendly as possible. In other words, from the available materials, a better material was chosen based on the stress analysis of the vessel (using the finite element approach). The numerical analysis of the vessel was carried out by S.A. Lathuef and C.K. Sekhar [8] with a view to optimize the design of vessel.

Khobragade and et.al. [9] accomplished by altering the shell's thickness and nozzle placement without sacrificing strength. The ideal thickness for the vessel and nozzle position were suggested. The vessel's tension is reduced by positioning the nozzle on the dish end with the slope facing up. To determine the impact of the number of saddle supports for hemispherical and flat heads on the stress distribution, a different study of the pressure vessel was conducted using finite element software. The vessel with a hemispherical head and three saddle supports was shown to be the least strained.

M. Jadav Hyder and M.Asif [10] optimize the location and size of opening in pressure vessel cylinder using ANSYS, Analysis performed for three thick-walled cylinders with different internal diameter. They started by analysing pressure vessel cylinders without holes and discovered tangential, longitudinal, radial, and von Misses stresses. Next, they optimised hole sizes by creating holes of varying diameters in the centres of each of three thick cylinders. They discovered that the best hole size is 8 mm for cylinders with internal diameters of 20 cm, and that holes of 10 mm and 30 cm have the lowest von Misses stress values and finally 12 mm hole located at 1/6, 1/8, 2/8, 3/8, and 4/8 of cylinder from top in all three cylinders. Finally, they discovered that the Von Misses stress is minimal at the location 1/8 of the cylinder height. They discovered that the Von Misses stress is maximum at the centre 0.500 location, decreased directed away from the centre, and increased at the location change from 0.1250 to 0.0625 from the cylinder top due to end effect.

Two different nozzle geometries are being investigated by Joship Kacmarcik and Nedelijko Vukojevic [11], and two different methods, strain gauge with experimental set-up and finite element analysis with ABAQUS software, are being used. Here, two stress concentration factors defined by maximum principal stress and maximum von Mises stress are calculated by strain gauge measurement and compared with ABAQUS software. This comparison shows good agreement between the stress concentrations factors determined with two different methods. Because it is feasible to specify three symmetry planes, only one-eighth of the vessel and one-fourth of the nozzle are modelled in this study. C1 Nozzle has a larger radius than C2 Nozzle, but both nozzles have the same thickness of the vessel wall and exterior radius. When comparing the two methods, it can be seen that the maximum deviation of 15.5 percent is acceptable for engineering applications of stress concentration factor and that FEM analysis is extremely reliable enough for determining stress concentration factor in pressure vessel design. Here, 3D tetragonal elements are implemented as a mesh generation method, and stress concentration factor is obtained by the value of stress (principal and von Mises) obtained via FEM analysis and strain gauge measurement. Additionally, this study demonstrates the benefits of using FEM analysis to identify stresses on a vessel's internal side that may be higher than external stresses and are very challenging to quantify with strain gauges.

Using the Timoshenko shell theory and the finite element method, V.N. Skopinsky and A.B. Sinetankin [12] presented work on modelling and stress analysis of nozzle connections in Ellipsoidal heads of pressure vessels under external loading. The effect of stress concentration in external loading has more effect than in the internal pressure, and there is a noticeable increase of the maximum stress for shell in the interaction region even at the small level of nominal stress. The interaction curve between the nozzle and the head has a non-uniform distribution of stress due to non-radial and offset connections; this paper shows the influence of angular parameter for non-radial nozzle connections. Maximum effective stress decreases as angle increases more significantly for non-central connections, and in the case of torsional moment loading, the angle affects the stress in the opposite way, with the stress in the shell increasing as alpha angle increases.

Three pairs of full-scale test vessels with different mean nozzle to mean diameter of cylindrical vessel ratios were designed and fabricated for testing and analysis, were done by Fang et.al. [13]. They presented work on a comparative study of strength behaviour for cylindrical shell interaction with and without pad reinforcement under out-of-plane moment loading on nozzle. The material of the cylinder, reinforcement pad, and nozzle is low carbon steel. They discovered that the plastic limit of the nozzle in the cylinder vessel is increased by pad reinforcement, with a general rate of increase of about 40-70 percent from test and it’s larger than 40 percent from finite element analysis. They also discovered that the rate of reduction depends on the structure and dimension of the vessel, for example, D/d ratio, and that the rate of reduction in test is 20–60 percent and in finite element analysis is 28–59 percent.

Pravin Narale and P.S. Kachare [14] presented work on structural analysis of nozzle attachment on pressure vessel design. If the nozzle is kept on the peak of the dished end, it does not disturb the symmetry of the vessel; however, if it is placed on the periphery of the vessel, it may disturb the symmetry of the vessel. Even within a single pressure vessel, the size, diameter, angle, etc. of the nozzle connection can vary significantly. These nozzles result in geometric discontinuity in the vessel wall, which creates a stress concentration around the opening. High stress can cause the junction to fail, so a thorough analysis is necessary.

The work on local pressure stress on lateral pipe-nozzles with various angles of interaction was completed by Xu et al. [15]. This paper reports variation of the local pressure stress factor at the pipe-nozzle junction when longitudinal stress at four symmetric points around the pipe nozzle junction are plotted as a function of an angle. The true pipe nozzle geometry was modelled using the ALGOR finite element programme. The numerical stress result comes from parameters beta and gamma, which are the nozzle mean radius and pipe thickness. At an angle of 90 degrees, the result had low local stress; however, as the angle of interaction decreases from 90 degrees and the angle decreases from 45 degrees, the stress value increases and the inside crotch point B has the worst circumferential stress. Additionally, it was less stressed than other angles.

Arman Ayobstress [16] studied the stress analysis of a toruspherical shell with a radial nozzle; in this paper, experimental readings were taken using a 0.0625-inch foil string gauge that was attached to the shell's outer and inner surfaces. The model was equipped with 39 pairs of 0.0625-inch foil strain gauges, which were spaced 39 pairs apart along the meridional axis. The experimental finding presented in this study comes from a test programme performed by drabbles to ascertain how a toruspherical vessel with a nozzle behaves during shakedown under the influence of internal pressure, thrust, and bending moment. The maximum stress could occur at any of the sphere-nozzle, sphere-knuckle, or cylinder-knuckle junctions, according to the graph of the elastic stress factor distribution along meridian plane due to the four load cases shown in this paper. The crotch corner and the weld-crown region are the areas of greatest stress, with an ESF of about 2.

V.N Skopinsky [17] had worked on stresses in ellipsoidal pressure vessel heads with non-central nozzle. The goal of this paper is a deeper investigation of the shell intersection problem. The shell theory and finite element method are used for stress analysis of nozzle connections in ellipsoidal heads of the pressure vessel. Here, nozzle is significantly displaced on ellipsoidal head from head axis is considered in this paper and structural modelling of nozzle-head shell intersections and SAIS special-purpose computer program are discussed. Under internal pressure loading, the results of a stress analysis and parametric study of an ellipsoidal vessel head with a non-central nozzle are shown. In many real-world designs, the nozzle is positioned a fair distance from the head axis. This stress analysis specifically takes into account these cases, which leads to a better understanding of this understudied issue and the potential for more reliable nozzle connection designs on pressure vessel heads. Additionally, the SAIS programme can be used for design optimization purposes, such as nozzle location determination.

Jaroslav Mackerle [18] worked on a bibliographical review of the finite element method (FEMs) used for the theoretical as well as practical analysis of pressure vessel structural/components and piping. He searched a paper that contained 856 references to papers and conference proceedings on the topic that were published in 2001–2004 and discovered papers that are categorised in the following categories: Static and dynamic, stress and deflection analysis, contact problem, fracture mechanics problem, stability problem, heat problem, fracture mechanics analysis problem He discovered that there were many themes in pressure vessel and piping that were related to linear and nonlinear, static and dynamic, static and deflection analysis, and fracture mechanics problems. He developed specific finite elements for pressure vessel and pipes as well as other topics.

Balicevic et al. [19] presented work on the ANALYTICAL and NUMERICAL solution of internal forces by cylindrical pressure vessel with semi-elliptical heads; in this paper, the solution for internal forces and displacement in the thin-walled cylindrical pressure vessel with ellipsoidal head has been proposed using the general theory of thin walled shell of resolution and distribution of the forces and displacement in thin walled shell are given in mathematical form. To confirm the analytical solution, a finite element analysis of a cylindrical vessel with a semi-elliptical head was conducted using the ANSYS 10 code. Here, the ellipsoidal head model was created as an axe-symmetric problem to prevent bending effects on the contact between heads and cylinders, and the author concluded that the principal stresses calculated analytically are very close to the finite element results, the difference is less than 3%.

Hsieh et.al. [20] *had* worked on nozzle in the knuckle region of a torispherical head. In this paper, limit load interaction plots for pressure versus nozzle axial force, in-plane moment, out-of-plane moment, and for in plane moment versus out-of-plane moment are also present. Here, six models are included with nozzle offset location nozzle offset/vessel outer diameter in the present study, with model 1 being the asymmetric case with nozzle located in the centre of the crown. The model 3 offset outermost weld location is at crown/knuckle junction. The FE model was created using the PATRAN mesh generation programme, and stress analysis work was completed using the ABAQUS programme. They came to the conclusion that the nozzle has very little influence on the limit pressure of the head, even when it is located in the knuckle region of the head, and that for external loads applied to the nozzle, the effect of increasing the offset is to increase the limit loads.

The pressure vessel was designed utilising ASME regulations and standards by B.S. Takkar and S.A. Takkar [21] in an effort to legitimate the design. A set of tests to the applicable ASME standard in the future scope they have stated can be used to establish how well a pressure vessel performs under pressure. PVELITE software allows for the design of pressure vessels. Additional FEA analysis can be performed to confirm the design process described above. They came to the conclusion that designing a pressure vessel is more of a selection process than a designing of every component. Pressure vessel components are chosen based on the ASME standards that are currently in effect, and manufacturers also adhere to these standards when producing the components, freeing the designer from designing the components. This feature of the design greatly reduces the amount of time it takes to develop a new pressure vessel. It also enables the designer to avoid creating multiple pressure vessel prototypes before settling on the final design. In addition, because standard parts are used, the cost of the design is lower overall.

Governing shell thickness in accordance with ASME may have various unforeseen implications, as discussed by Shaik Abdul Lathuef and K. Chandra Sekhar [22]. Here, there is room to alter the code values by increasing the minimum governing thickness of the pressure vessel shell to the necessary standards and moving the nozzle placement to reduce shell stresses. In this work, the nozzle is positioned at five different locations and is analysed using ANSYS. It is located at the left end of the shell, the centre of the shell, the right end of the shell, and the dished end on both sides. The stress would be at a minimum at the dished end with a hillside orientation, they discovered from the results. A low value for the safety factor results in material economy, which makes vessels thinner, more flexible, and more cost-effective. Here, we used the Zick analysis technique to evaluate the vessel's stress.

According to Binesh P. Vyas and R. M. Tayade [23], utilising PVElite for pressure vessel design results in reliable analytical results and saves time. Using graphical software called PVElite, a vertical pressure vessel has been created. Some input factors, such as volume, internal diameter, design pressure (either inside pressure or external pressure), temperature, material, processing fluid, etc., are necessary for developing vertical leg supported pressure vessels. PVElite provides thickness of shell, thickness of head, height of head, thickness of nozzle, and manhole. PVElite calculates local stress in accordance with welding research council (WRC) 107. Further research is required to investigate environmental parameters such as earthquake, thermal load, fluctuation load, and so on.

Dražan Kozak and Ivan Samardžić [24] had worked on stress analysis of cylindrical vessel with changeable head geometry. The primary goal of this paper is to compare numerical and analytical results for pressure vessels with hemispherical heads using numerical analyses of cylindrical pressure vessels with changeable head geometry (semi-elliptical and hemispherical heads) and measures of precision and time needed to obtain the solution. This study uses SOLID 95, PLANE 183, and SHELL 181 components to perform a numerical analysis of a pressure vessel with hemispherical and semi-elliptical heads. The use of the PLANE 183 element, which requires the fewest meshing elements and the quickest calculation times, is determined to be the best strategy in both pressure vessel head cases. This type of asymmetric element may be suggested in such situations when the total symmetry of the model is taken into account.

Kumar et.al. [25] have done the analysis for different head shapes, different nozzle positions and variation in saddle supports. In their study, analytical software is used to build and optimise a horizontal pressure vessel. Based on the analysis's findings, recommendations are made for the ideal head shape, input and output nozzle placement, and support placement.

The aforementioned literature makes clear that pressure vessel designs were individually improved using numerical analysis for various circumstances. That is, just a small number of factors were taken into account, such as head form and supports, vessel materials alone, nozzle placement and thickness, or head shape and shell thickness. All these factors must be taken into account throughout the optimization process for a better design. Therefore, in this work, the design optimization of a horizontal vessel is carried out by numerical analysis while taking into account various head forms, the placement of the nozzle, and the quantity and location of supports. The ASME code may be used to determine the minimum shell thickness necessary to resist the operating loads. Similar to this, the right material may be chosen based on its availability and qualities and be able to safely handle the imposed load. Therefore, the optimization method does not take these two factors into account.

2.3 Gap in literature review and problem identification

Shell thickness calculation to be done with other competing materials. Rigorous analysis for torispherical, ellipsoidal and hemispherical to be carried out (Design of rules and design by analysis). Nozzle reinforcement to be detailed (Design by rule in particular). More configuration for pressure vessel support system can be tried. Joint efficiency to be considered as per ASME BPV code in further calculations.

Therefore, in this work, the design optimization of a horizontal vessel is carried out by numerical analysis while taking into account various head forms, the placement of the nozzle, and the quantity and location of supports. The ASME code may be used to determine the minimum shell thickness necessary to resist the operating loads. Similar to this, the right material may be chosen based on its availability and qualities and be able to safely handle the imposed load. Therefore, the optimization method does not take these two factors into account.

2.4 Objectivity

The main objective of this thesis work is to minimize the total stress and total deformation of a Pressure vessel subjected to stress constraints specified by the ASME 'Design by Analysis of Boiler and Pressure Vessel' code limits. This is achieved by using FEA results obtained from ANSYS in conjunction with CATIA for the sole purpose of minimizing the objective. The subject of design optimization has been expanded by the ability to solve complicated design issues by combining reliable tools like CATIA with potent FEA programmes like ANSYS. The experimental process of conceptual and detailed engineering system design can be enhanced by optimization techniques paired with more precise and thorough simulation techniques. The post-processing findings are saved in a text file and include the total volume of the apparatus, deformation, von-Mies stress, linearized membrane, and membrane plus thermo mechanical stresses. By using the necessary information from this result file produced by ANSYS, the objective function and restrictions are constructed in CATIA. While the linearized stress findings, which are checked against their ASME permissible values, comprise the constraint, the volume data provides the value of the objective function. Finally, the ANSYS compares numerous solutions until the best design is discovered before repeatedly evaluating the goal function. Up until the ideal answer is discovered, the suggested process is fully automated and doesn't need any form of user input.

CHAPTER 3: FINITE ELEMENT ANALYSIS OF THE PRESSURE VESSEL

3.1 Modelling of Horizontal Cylindrical Pressure vessel using CATIA

The pressure vessel is developed in accordance with the guidelines and requirements detailed in the machine design and design data book. Dimensions are computed and employed in the pressure vessel modelling seen in Fig. 3.1.

Our design productivity will increase with a mechanical design solution. This software is a collection of applications used in the design, analysis, and production of an almost infinite variety of products. It implies that rather of providing low level geometry like lines, regions, and circles, we design components and assemblies by defining features like extrusion sweeps, cuts, holes, and so forth. This enables the designer to conceptualise the computer model at a very high level while leaving CATIA to handle any low-level geometry details.

The CATIA sketcher Menu bar/tools icon is shown. It provides a greater flexibility in editing, for example, if we have to change the dimensions in design assembly, manufacturing etc. modifications will automatically propagate in other modules of CATIA. It provides efficient 3-D Models which are easy to visualize in other compatible software and also it can be transferred for assembly, FEA and CNC machine. It also reduces the time required for complete design and manufacturing of the component to a large extent.

Here, Horizontal cylindrical pressure vessel is modelled in three body parts and then assemble all the bodies using Boolean operation in part design. The body parts like saddle support, cylindrical shell with heads and nozzles are shown.

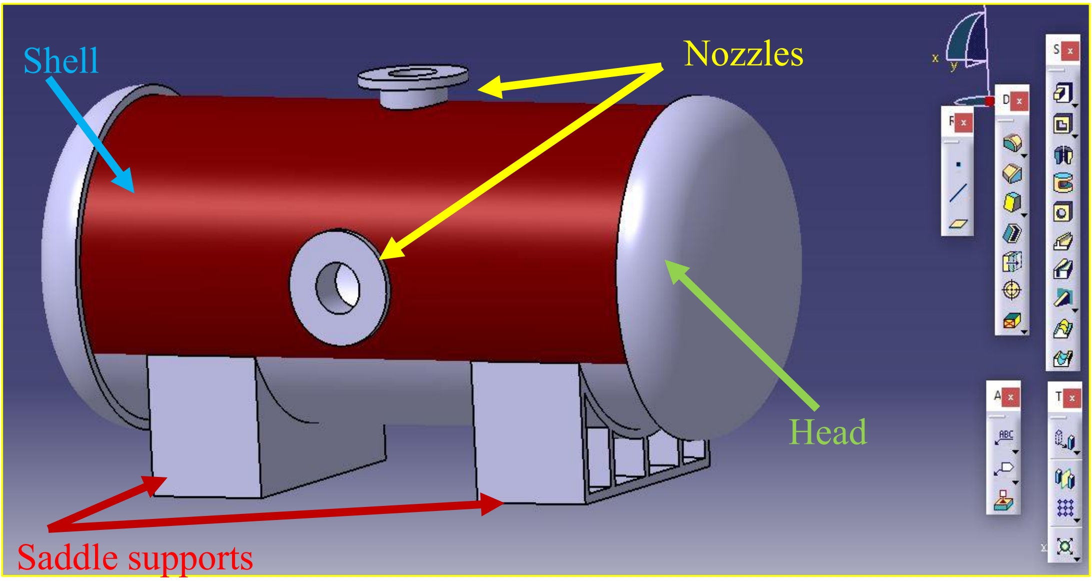


Fig. 3.1: CATIA model of horizontal pressure vessel depicting shell, nozzles, heads and saddle supports

The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material. Fig 3.2 shows CATIA model of horizontal pressure vessel having different head shapes. (a) Ellipsoidal, (b) Hemispherical and (c) Torispherical are shown below.

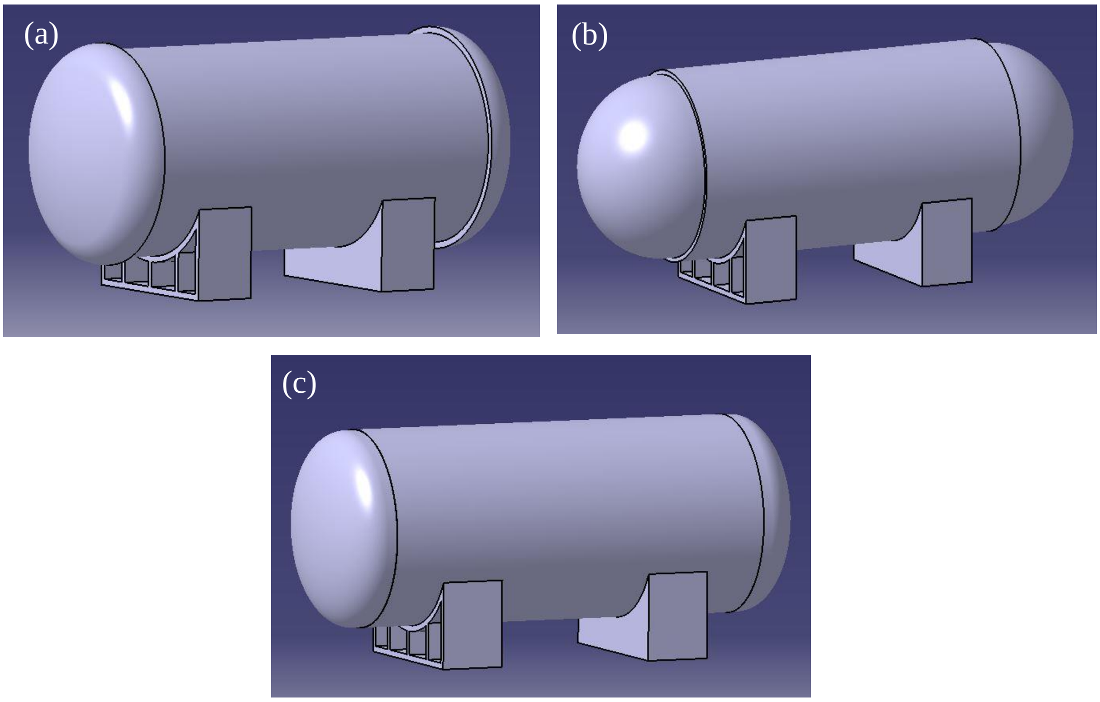


Fig. 3.2: CATIA model of horizontal pressure vessel having head shapes shown in (a) Ellipsoidal, (b) Hemispherical and (c) Torispherical

Three combinations of nozzle positions are chosen as inlet and outlet. Fig 3.3 shows CATIA model of horizontal pressure vessel having different nozzle positions with torispherical head. Inlet and outflow nozzles are situated at the top and side of the shell, respectively, 25 percent of the length of the shell from one end, as illustrated in fig. 3.3 (a). Fig. 3.3 (b) depicts the outlet nozzle on the head and the inlet nozzle at a distance of 25% of the shell's length from one end. Fig. 3.3 (c) depicts the inlet and outlet nozzles at the top and side of the shell, respectively, at a distance of 50% of the shell's length from one end, or in the shell's middle.

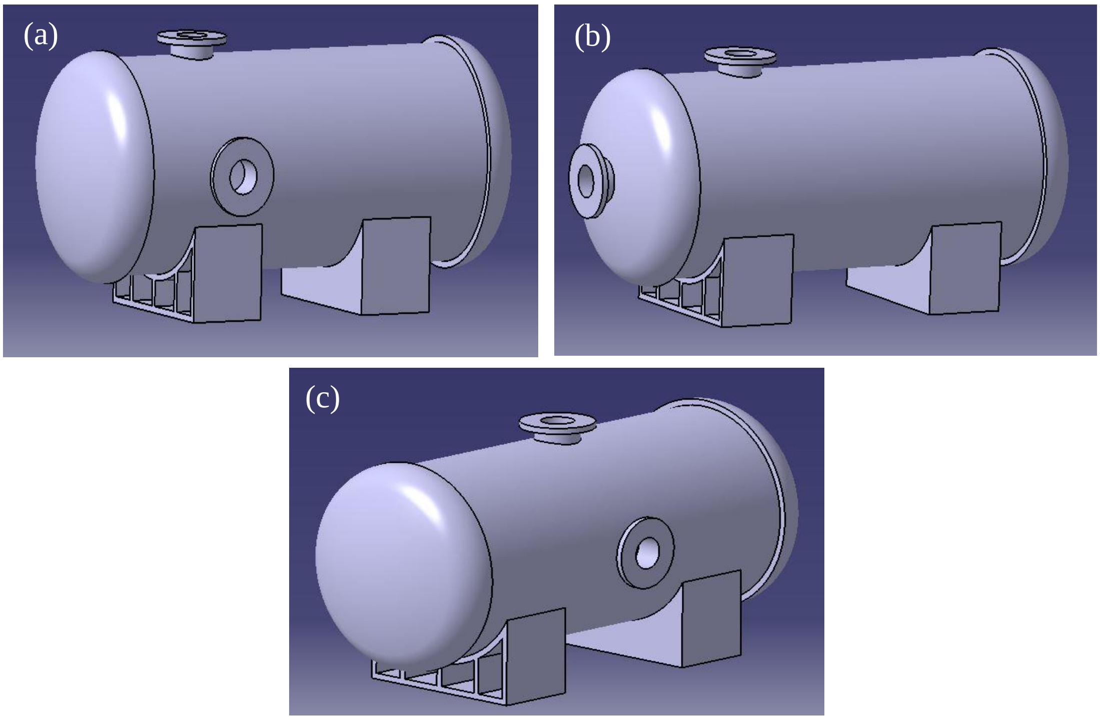


Fig. 3.3: CATIA model of horizontal pressure vessel having different nozzle positions with torispherical head shown in (a) nozzle located at top and side of the shell at a distance of 25% of total length of shell (b) outlet nozzle on the head and inlet nozzle located at a distance of 25% of total length of shell (c) inlet and outlet nozzle located at the centre of the shell

The supports value is varied from 1m to 2m. Two saddle supports are chosen and the effect of distance between them is examined which shows in CATIA model of horizontal pressure vessel in fig.3.4. The three cases considered in this analysis are shown in fig 3.4 (a) 1 m distance between centre of shell of and centre of the support. Fig 3.4 (b) shows 1.5 m distance between centre of shell of and centre of the support and fig 3.4 (c) shows 2 m distance between centre of shell of and centre of the support.

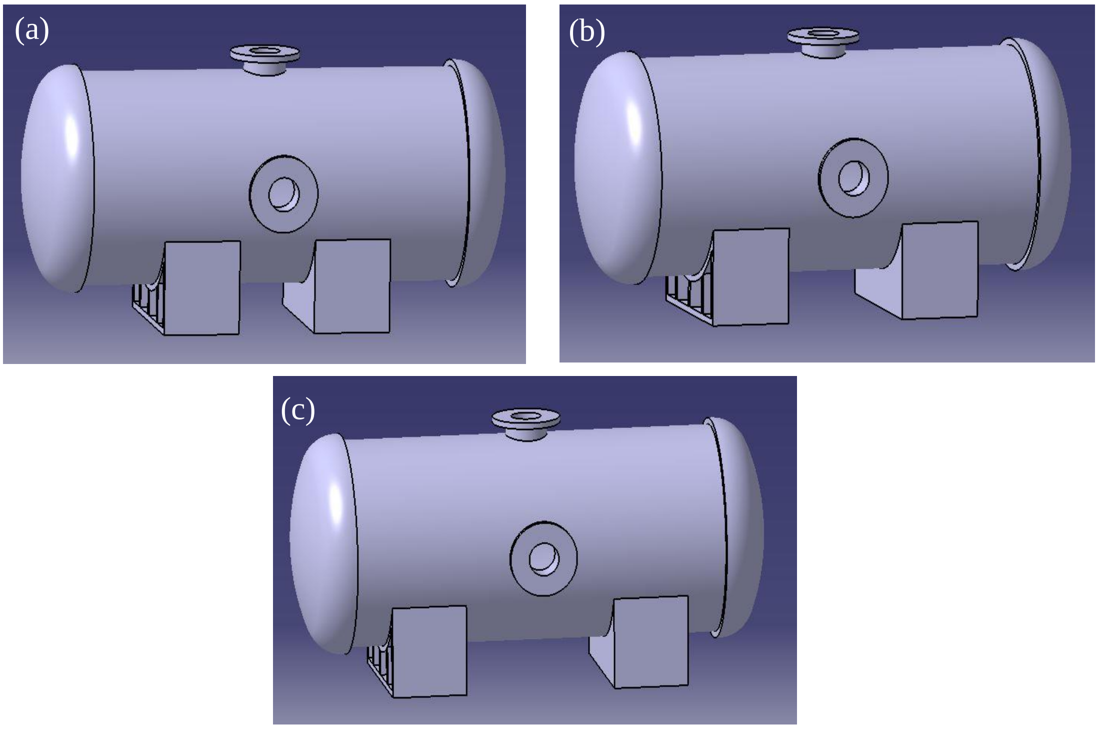


Fig. 3.4: CATIA model of horizontal pressure vessel having different saddle supports positions with torispherical head shown in (a) 1 m distance between supports (b) 1.5m distance between supports (c) 2 m distance between supports

3.2 Static structural Analysis of Model through ANSYS workbench

A 3D model of the vessel geometry must be built in order to analyse the pressure vessel in ANSYS Workbench. It is necessary to create a 3D model of both thin and heavy wall configurations. To provide results that closely match the actual structural behaviour under the applied loads, the model should be as precise as is practical. The pressure vessels are displayed in a 3D model. The heavy wall configuration is displayed at the bottom, while the thin wall configuration is displayed at the top. CATIA V5R20 is used to create the 3D models. All the CATIA V5R20 models must be imported to ANSYS R19.2 so that further analysis will be performed.All the ANSYS 3-D model are shown below.

The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material. Fig 3.5 shows CATIA model of horizontal pressure vessel having different head shapes (a) Ellipsoidal (b) Hemispherical and (c) Torispherical are shown in figures respectively in Fig 3.5 (a), (b) and (c).

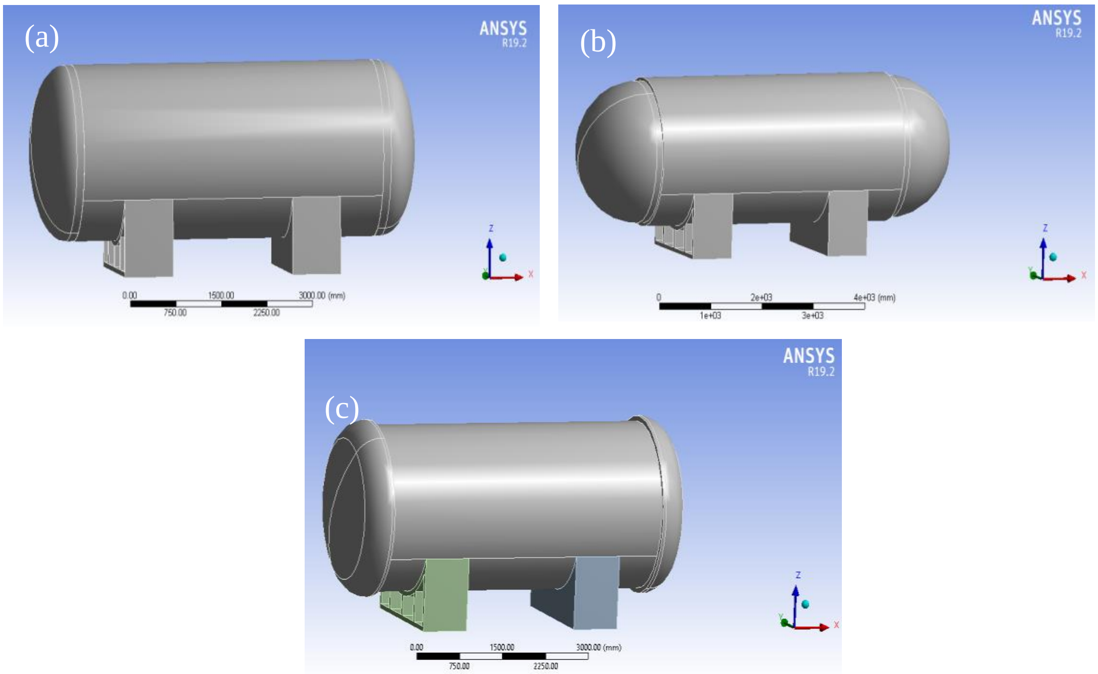


Fig. 3.5: ANSYS 3-D model of horizontal pressure vessel having different head shapes shown in (a) Ellipsoidal (b) Hemispherical and (c) Torispherical

Three combinations of nozzle positions are chosen as inlet and outlet. Fig 3.6 shows ANSYS 3-D model of horizontal pressure vessel having different nozzle positions with torispherical head.

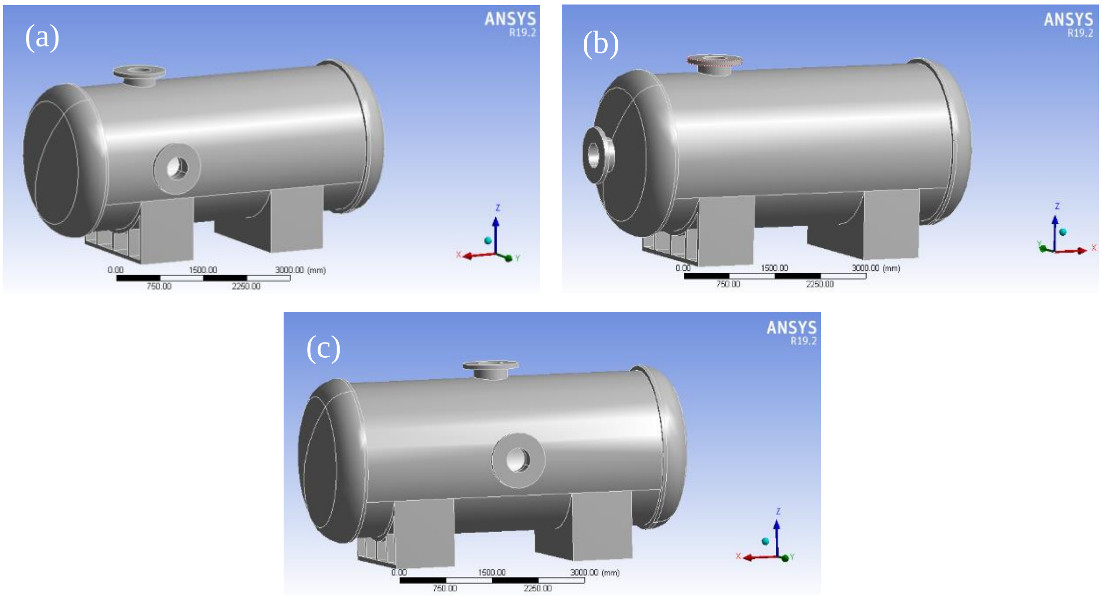


Fig. 3.6: ANSYS 3-D model of horizontal pressure vessel having different nozzle positions with torispherical head shown in (a) nozzle located at top and side of the shell at a distance of 25% of total length of shell (b) outlet nozzle on the head and inlet nozzle located at a distance of 25% of total length of shell (c) inlet and outlet nozzle located at the centre of the shell.

Fig. 3.6 (a) illustrates the location of the input and outflow nozzles, which are situated at the top and side of the shell, respectively, and spaced 25 percent of the shell's length apart. Fig. 3.6 (b) depicts the outlet nozzle on the head and the inlet nozzle at a distance of 25% of the shell's length from one end, respectively. Fig. 3.6 (c) depicts the inlet and outlet nozzles at a distance of 50% of the shell's length from one end, i.e., at the shell's middle.

The supports value is varied from 1m to 2m. Two saddle supports are chosen and the effect of distance between them is examined which shows in ANSYS 3-D model of horizontal pressure vessel in fig.3.7. The three cases considered in this analysis are shown in fig 3.7 (a) 1 m distance between centre of shell of and centre of the support. Fig 3.7 (b) shows 1.5 m distance between centre of shell of and centre of the support and fig 3.7 (c) shows 2 m distance between centre of shell of and centre of the support.

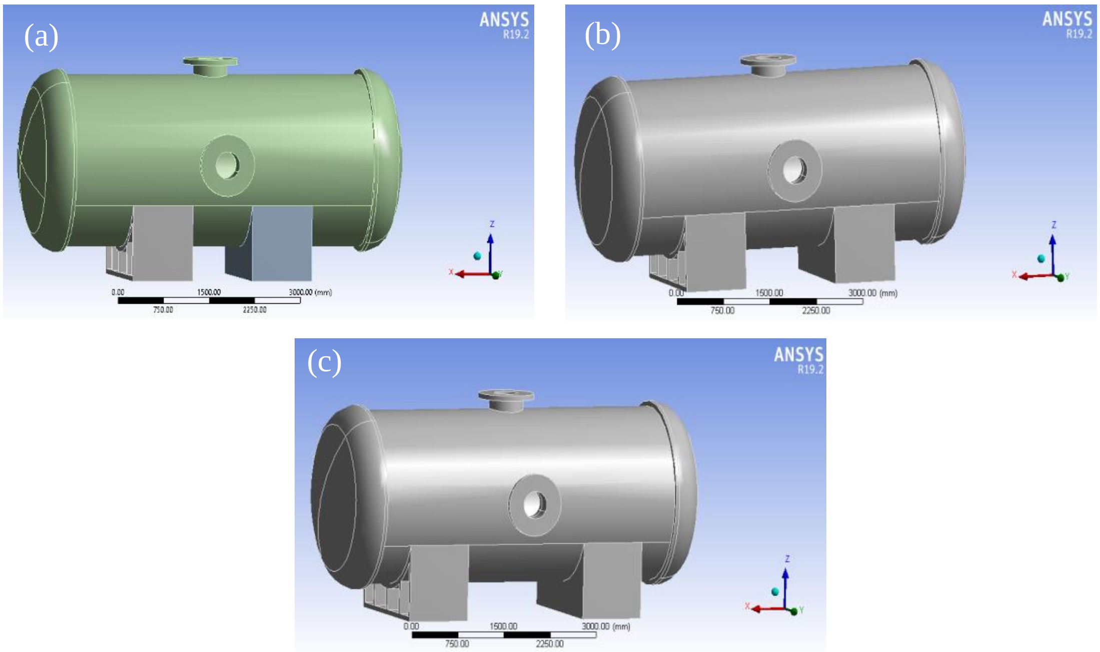


Fig. 3.7: ANSYS 3-D model of horizontal pressure vessel having different saddle supports positions with torispherical head shown in (a) 1 m distance between supports (b) 1.5m distance between supports (c) 2 m distance between supports

3.3 Boundary and loading conditions:

Different boundary conditions are imposed on the design model: (1) Symmetry boundary condition is applied to the structural symmetry plane.

(2) Saddle supports are fixed in such a way that fixed support in X-axis and Y-axis on the position of one saddle and Fixed support in X-axis on the position of another saddle which is shown in Fig.3.8

Different loading conditions are applied on the pressure vessel model.

(1) Acceleration due to gravity along z axis.
(2) An Internal pressure of 10 MPa is applied to inner surface of the cylinder and head as shown in Fig.3.9.

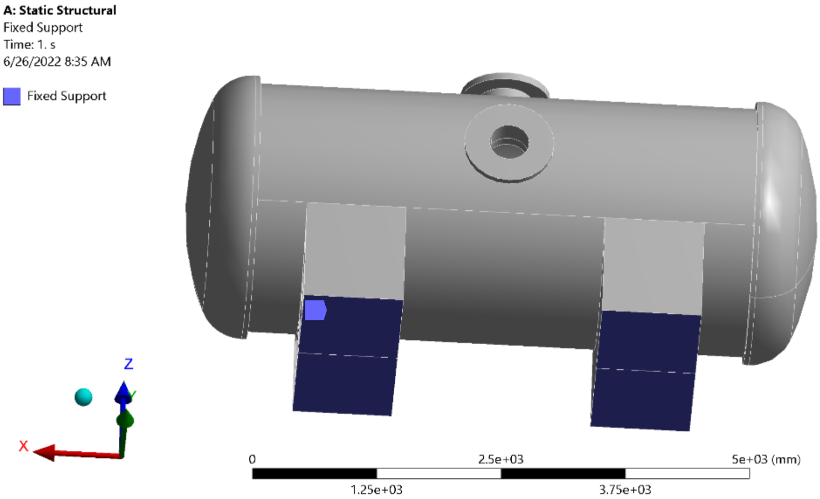


Fig. 3.8: Fixed saddle supports

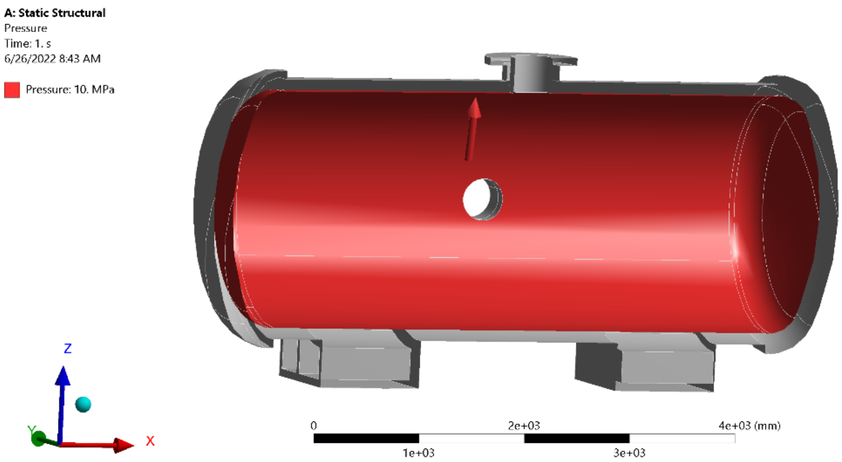


Fig. 3.9: Sectional view of HPV having internal pressure of 10MPa at inner surface

3.4 Meshing

Meshing is one of the most important aspect of finite element analysis. The accuracy of the FEA results predominantly depends on the mesh size and element quality. The larger the density of meshing, the greater is the accuracy of the geometry and greater is the difficulty in solving the problems. Therefore, a preferred meshing approach is to employ fine meshes only in the area of focus whereas larger meshes should be used in the region where we expect relatively low activity. The pattern and relative positioning of the nodes also affect the solution, the computational efficiency & time. This is why good meshing is very essential for a sound computer simulation to give good results.

For the pressure vessel model, besides four volume block, the entire model was meshed with 20- node Brick elements. Since these four volumes does not meet the hexa-meshing criteria, they were meshed with 10-node tetrahedral elements. The ANSYS software automatically uses pyramid elements as filler elements in between the mesh transition zones. The Hexahedral meshed model of the pressure vessel used during optimization is shown below in figure. All the meshed 3-D model are shown below.

The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material. Fig 3.10 shows meshed model of horizontal pressure vessel having different head shapes (a) Ellipsoidal (b) Hemispherical and (c) Torispherical are shown in figures respectively in fig 3.10 (a), (b) and (c).

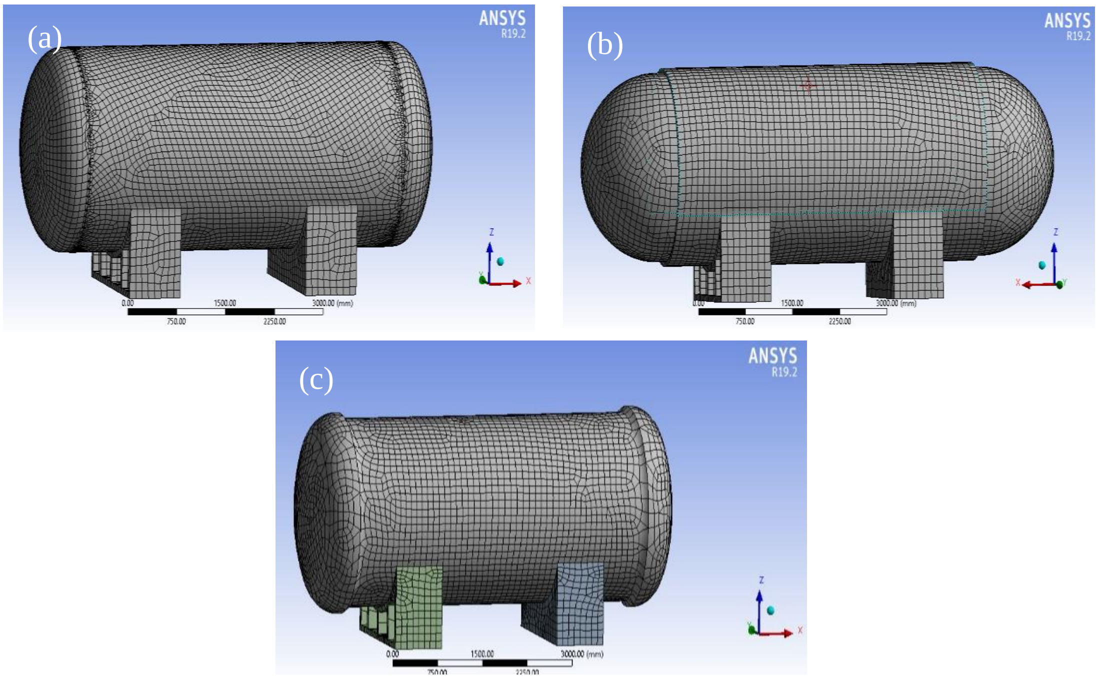
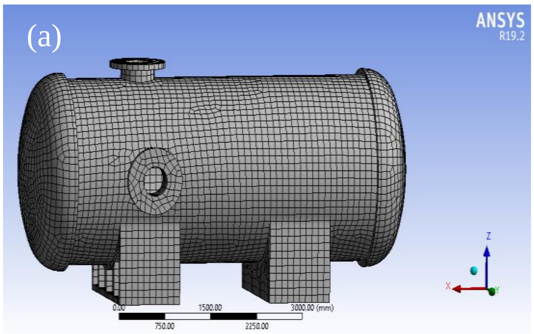
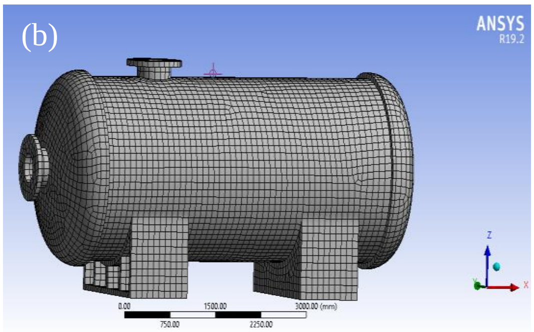


Fig. 3.10 Meshed model of horizontal pressure vessel having different nozzle heads shown in (a) Ellipsoidal (b) Hemispherical and (c) Torispherical

Three combinations of nozzle positions are chosen as inlet and outlet. Fig 3.11 shows meshed 3-D model of horizontal pressure vessel having different nozzle positions with torispherical head. Inlet and outlet nozzles are situated at the top and side of the shell, respectively, 25 percent of the length of the shell from one end, as illustrated in Fig. 3.11 (a). Fig. 3.11 (b) depicts the outlet nozzle on the head and the inlet nozzle at a distance of 25% of the shell's length from one end. Fig. 3.11 (c) depicts the inlet and outlet nozzles at the top and side of the shell, respectively, at a distance of 50% of the shell's length from one end, or in the shell's middle.





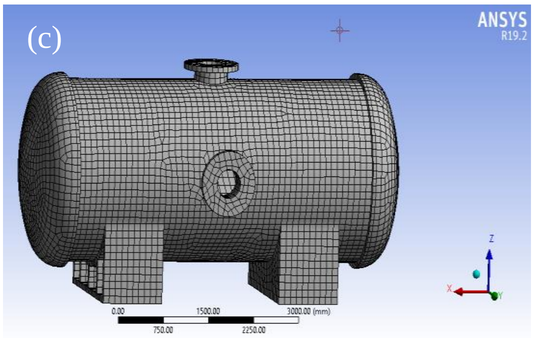


Fig. 3.11: Meshed model of horizontal pressure vessel having different nozzle positions with torispherical head shown in (a) nozzle located at top and side of the shell at a distance of 25% of total length of shell (b) outlet nozzle on the head and inlet nozzle located at a distance of 25% of total length of shell (c) inlet and outlet nozzle located at the centre of the shell.

The supports value is varied from 1m to 2m. Two saddle supports are chosen and the effect of distance between them is examined which shows in meshed model of horizontal pressure vessel in Fig.3.12. The three cases considered in this analysis are shown in Fig 3.12 (a) 1 m distance between centre of shell of and centre of the support. Fig 3.12 (b) shows 1.5 m distance between centre of shell of and centre of the support and Fig 3.12 (c) shows 2 m distance between centre of shell of and centre of the support.

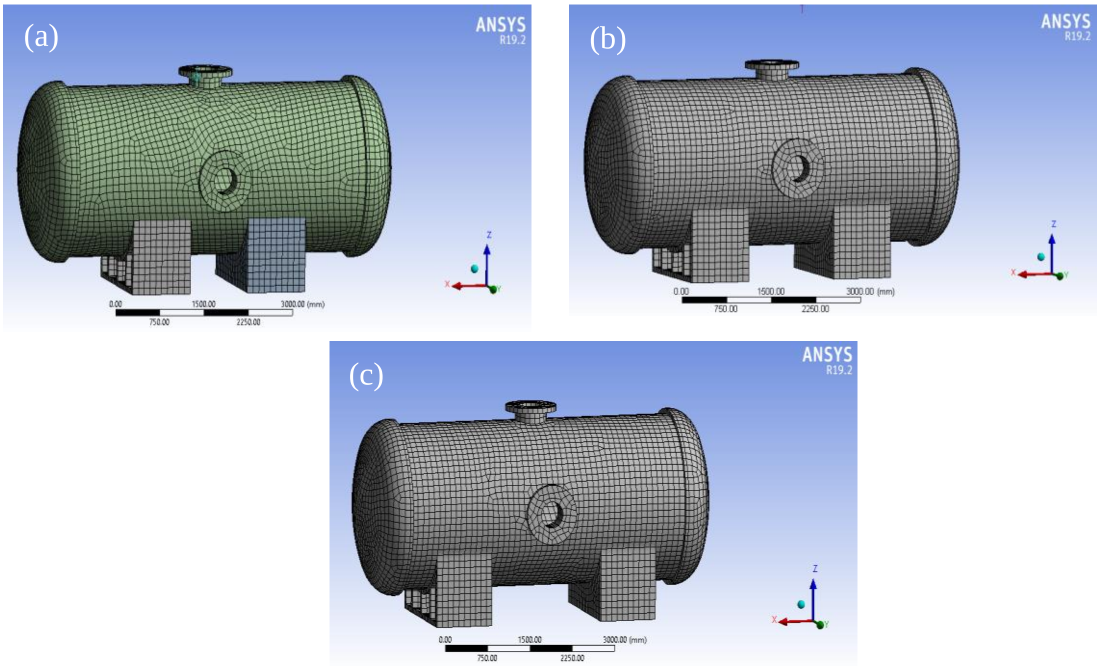


Fig. 3.12: Meshed model of horizontal pressure vessel having different saddle supports position with torispherical head shown in (a) 1 m distance between supports (b) 1.5m distance between supports (c) 2 m distance between supports

3.4.1 Tetrahedral mesh VS hexahedral mesh

Hex meshing (Brick elements) needs some topological cleaning and decomposition to produce an all or almost all brick mesh, in contrast to tetrahedron meshing, which can be carried out on almost any geometry. When fewer nodes and elements are needed yet high solution accuracy is required, this sort of meshing is typically used. In comparison to an all-tetrahedron mesh, a brick meshed model may frequently provide superior accuracy while using orders of magnitudes less CPU time, RAM, and storage space. But the downside of Brick meshing or a hexahedral mesh is that it is very difficult to generate for a complex geometry because it requires map mesh able sides to sweep through the volume.

A tetra mesh of the pressure vessel model having ten node tetrahedral elements were used to produce the tetra mesh model. For free meshing, a smart sizing level-2 was set to obtain a very fine mesh with better element quality. Whereas, a hex-mesh model was produced by sweep meshing the volumes. The total vessel volume was split into different blocks for more control and to meet topology requirement of map mesh able sides. The element sizes generated on swept volume were defined by assigning line divisions, taking into account curvature of the line, its proximity to holes, element order and other features. Below mentioned guideline was followed to achieve the volume sweep.

1. The source and target faces for all sweep able bodies are automatically detected by the ANSYS software. If desired, the user can specify the source/target faces manually.
2. All source/target face topology needs to be same for all sources/targets.
3. All side faces need to be able to be mapped meshed.

It is evident from the figure below, that the tetra meshed model contain eight times more elements compared to the number of elements in brick meshed model. The stress results obtained from both the meshes are more or less similar but the time taken to solve the tetra meshed model is about five times more when compared to the computational time taken to solve the brick meshed model.

3.4.2 Comparison between free mesh and controlled mesh

Meshing is one of the most important steps in performing an accurate simulation using FEA. A mesh is made up of elements which contain nodes (coordinate locations in space that can vary by element type) that represent the shape of the geometry. An FEA solver cannot easily work with irregular shapes, but it is much happier with common shapes like cubes. Meshing is the process of turning irregular shapes into more recognizable volumes called “elements”. During free mesh, line divisions are not considered and the software automatically determines the size and number of elements generated through the swept volumes. Whereas, the controlled mesh is created by specifying number of line divisions on the sweep able faces of the model geometry to produce different element sizes on different components of the pressure vessel. Latter is preferred as it gives more control over the mesh size and element quality. Note that, the free mesh generated by ANSYS, has less number of elements when compared to elements in controlled mesh model. Although controlled mesh increases the number of brick elements, the computational time taken to solve this model in a 16GB RAM system is approximately same when compared to the time taken to solve free mesh model. We also achieve a high solution accuracy due to the increased number of elements. Hence, controlled brick meshed model is preferred for the finite element analysis as well as during the optimization run.

3.4.3 Mesh convergence and stress singularity

A mesh convergence study when performing Stress Analysis is necessary to instil confidence in FEM results from the standpoint of mathematics. As we progressively refine the mesh, the size of the elements reduces, which theoretically increases the solution accuracy and given enough iterations it converges towards a specific result. The mesh refinement process will converge to the precise answer if there is an analytical solution for the given issue. The amount of computing needed to solve a particular model increases as the size of mesh components decreases but their number increases. Mesh components approach a threshold of diminishing returns in terms of precision as they get smaller relative to the computing burden and time needed to compute the result. This implies that a simulation takes significantly longer to compute the results, yet the outcome may vary by a negligible amount. Hence, to overcome this problem, mesh 26 convergence study is performed to determine a mesh with minimum number of elements required to maintain a satisfactorily balance between accuracy and computing resources.

However, as shown in table and figure below, the mesh convergence study when applied to the 20-node brick meshed pressure vessel model resulted in non-convergence. The stress solution does not converge with mesh refinement because of the stress singularity present at the joint corner of cylindrical shell body and nozzle.

An area of the mesh where the stress does not converge to a particular value is known as a stress singularity. The stress at this moment keeps rising as we continue to refine the mesh. The tension at the singularity is theoretically unbounded. The application of a point load, acute re-entrant corners, corners of bodies in contact, and point restraints are examples of circumstances where stress singularities commonly occur. These singularities occurs often in pressure vessel and boiler designs is practically unavoidable. Since the pressure vessel model is analysed as whole structure, the stress singularity at the filleted corner is of importance. The mesh around this region is refined locally to capture the effects of high stress concentration. Despite of removing sharp re-entrant edges by filleting the corners, the stress concentration around these corners increases with increase in the elements. However, displacement solution does converge to a certain value. Through path operations for critical stress concentration lines at stress singularity region, we can predict probable value of true stress at the elements near this singularity. While running the optimization loop, mesh density of the pressure vessel model was kept fixed and the stresses evaluated near the singularity were verified against the allowable local stress limits. The path operations and stress limits are discussed in the next chapter. The figures displayed below, shows the increase in maximum local un averaged von-misses stress with the increase in number of elements at the corner of the nozzle and shell body joint.

CHAPTER 4: STRESS, DEFORMATION ANALYSIS AND VERIFICATION

4.1 Introduction to ASME boiler and pressure vessel verification code

Relevant Codes of Practice, Industry Standard and/or Statement of Assessment Criteria: ASME Boiler and Pressure Vessel Code, Section VIII, Rules for Construction of Pressure Vessels, Division 2.

Boilers and pressure vessels are used worldwide in various industries. They are naturally present in the power engineering and gas engineering sectors. In order to ensure the safety and operational efficiency of these vessels, necessary legal regulations have been developed by the American Society of Mechanical Engineers (ASME). These regulations constitute the basis for design and manufacture of these equipment's. One of the most important goal in a pressure vessel design is to assure safe and satisfactory performance of the vessel. The ASME pressure vessel code is based on the observed safety of vessels. The observations were transformed into design guidelines, and as a result, the vessels were safer. It is a working standard that has origins in the failure of vessels and the deaths of operators from a time before ideas like stress concentrations were ever understood. The ASME 'Design by analysis' code, particularly in its Section VIII division-2, has specific requirements on how to assess the results from the stress analyses to make the necessary verifications to avoid failure. They wrote the VIII-2 rules and developed the stress linearization method as a guideline to check for the safe design. This division covers the mandatory requirements, specific prohibitions and non-mandatory guidance for materials, design, fabrication, inspection and testing, markings and reports, overpressure protection and certification of pressure vessels having an internal or external pressure which exceeds 10 MPa. These requirements apply to those equipment's which are part of the pressure boundary for example valves, pumps, pressure vessels, piping, etc. In this work, only the maximum allowable stress verifications were made as per the ASME code. Stress linearization method was implemented to verify the design of pressure vessel equipment under study. The details of the ASME codes are not included in this report. The reader is encouraged to the read the ASME related reference material mentioned at the end of this report.

4.2 Stress linearization

Generally, the prototypes and models used in the analyses are developed with 2D plane Or 3D solid finite elements, and membrane and bending stresses cannot be evaluated directly from the FEA results for these types of elements. Due to this fact, no direct Comparison with the code limits can be done and, besides that, the commercial finite Element software's like ALTAIR, ANSYS, ABAQUAS, etc do not distinguish between primary and secondary stresses. Therefore, to implement the required ASME Pressure Vessel and Boiler Code stress verifications for our finite element model, we should perform stress linearization to extract the membrane and bending stresses from the 3D solid model and, also should classify these stress components as primary and secondary for the purpose of stress verification against the ASME allowable limits. Linearization is a decomposition of the stress distribution we see in FEA of pressure vessels. It decomposes a basically parabolic distribution into a uniform value (membrane stress), a linearly varying value (bending stress), and possibly an extra component (peak stress). The stress linearization is performed along a stress classification line. The stress classification line (SCL) are created to linearize the stresses along a line, usually cutting through the thickness of the component. A Stress Classification Line or SCL is a straight line defined by two nodes/points, usually more or less perpendicular to both the inside and outside surfaces. Stress components through the section/SCL are linearized by a line integral method and are separated into constant membrane stresses, bending stresses varying linearly between end points, and peak stresses (defined as the difference between the maximum and minimum principal stress). The stress linearization tool takes the nodal data for the complex stress pattern found along this line and breaks it down into membrane and bending stress components. Stress Linearization is used to comply with design codes and requirements of the pressure vessel industry. However, applicability of the utility is not limited to pressure vessels. You can use this method to graph local stress tensors along a linear path and/or to determine the relative contributions of bending and membrane stress for any type of structure. Stress linearization is not required in the models with beam or shell elements because these elements naturally give the stresses separated in membrane and bending components.

Note:

• Stress linearization is available for brick, tetrahedral, plate, shell, and 2D elements, with or

Without mid-side nodes.

• Stress linearization is available for all linear and nonlinear analysis types that produce stress results.

• Stress Classification lines are created as paths in ANSYS.

4.3 Stress classification

The purpose of stress classification is to identify the Primary (P) and the Secondary (Q) stresses. Primary stresses are defined as the stresses developed by an imposed loading that is necessary to satisfy the laws of equilibrium in terms of the external and internal forces and moments. Secondary stresses are the stresses that are developed by constraints due to geometrical discontinuities and self-constraint. When stresses are divided into primary and secondary categories, the issues pertaining to overall strength which are of primary importance and are therefore classified as primary stresses are distinguished from those pertaining to local behaviour, which are of secondary importance and are classified as secondary stresses. It should be acknowledged that different kinds of stress have different degrees of significance and thus should have different safety implications. For example, the objective of primary stress limits is to prevent the loss of load-carrying capacity of the vessel, which is referred to as collapse whereas type of failure that a secondary stress may cause is ratcheting or incremental collapse.

Hence, it is necessary to classify the stresses into different categories. The stress categories of interest for the design analysis of our pressure vessel model are the primary stress, and its subcategories of general and local primary membrane and bending stress, and the secondary stresses. The peak stress is related to the assessment of fatigue failure of the material and will not be used in our analysis. For design purposes, the primary membrane stress is further divided into general primary membrane stress and local primary membrane stress subcategories. The average value acting on the whole section/line that is equivalent to the net force acting in the section due to the actual stress distribution will be classified as P m or PL depending on the distance of the section from the discontinuity: Pm for those far sections and PL otherwise. This PL classification is justified because there is a secondary 'aspect' in this stress near a discontinuity even if it comes from a mechanical load. The maximum value of the linear stress distribution which produces a net bending moment equivalent to the moment produced by the actual stress distribution is called 'bending stress' Pb. For mechanical loads, if the section is near a discontinuity this stress component is classified as secondary, 'Q'. The difference between the actual stress distribution and the sum of the average and linear (membrane + bending) stress distributions give an equilibrated stress distribution.

Pm – Generalized Primary Membrane Stress.

PL – Localized Primary Membrane Stress

Pb – Primary Bending Stress

F – Peak Stress

Q – Secondary Stresses

These steps, the stress classification and the stress linearization, are not straightforward ones and needs some 'engineering' judgment to choose the right section to evaluate the stresses in discontinuities. This task, most of the time is not a simple one due to the nature of the involved load and/or the complex geometry under analysis. In fact, there are several studies discussing on how to perform these stress classification and linearization.

4.4 Design limits and verifications

ASME Section VIII-2 provides a guide to what the maximum stresses are allowed for different locations of the pressure vessel. The combination of this ASME code and the output from the stress linearization and classification tool is used to produce pass fail judgments on the pressure vessel model. This will form the basis of constraint function in the optimization process. As the ASME limits are developed aiming to prevent some typical failure modes besides the Primary and Secondary classification, the stresses should be linearized to obtain the generalized (Pm) or localized (PL) membrane component, the bending (Pb) and the Peak (F) stress. Because different modes of failure are associated with primary membrane, primary bending and secondary stress, different allowable values are defined for each category. These are not given as absolute values in the pressure vessel codes, but as a proportion of the basic allowable stress intensity of the material (Sm) at design/working temperature. For the pressure vessel model in hand, standard steel S30408 was used, which has a basic allowable stress intensity value of 137 MPa at the working temperature of 20°C - 150°C.

Five Basic Stress Categories used for code verification are:

1) General Primary Membrane Stress Intensity (Pm)

2) Local Primary Membrane Stress Intensity (PL)

3) Primary Membrane Plus Primary Bending Stress (PL + Pb) – either General or Local Membrane Stress 34

4) Primary plus Secondary Stress Intensity (PL + Pb + Q)

5) Peak Stress Intensity (PL + Pb + Q + F)

According to the ASME code, the maximum allowable stress limits for the pressure vessel model in consideration are shown below:

CHAPTER 5: PARAMETRIC OPTIMIZATION

5.1 Optimization problem

Optimization may be described as the overall process of looking for the best design that is currently possible. By controlling a group of variable factors that have an impact on both the objectives and the design constraints, mathematical optimization is the process of maximising and/or decreasing one or more objectives while maintaining the established design constraints. It's crucial to understand that in order to use mathematical optimization, the objective(s) and design constraint(s) must both be expressed as quantitative functions of the variable parameters. Design variables and choice variables are other names for these variable elements.

Objective function: The objective function to be minimized is the maximum von misses stress and total deformation of the pressure vessel. Like most of the conventional optimization problems, this is also a single objective optimization problem. Instead of the conventional method where the objective evaluated as a function of design parameters, a scalar value representing the total volume of the pressure vessel structure, obtained from ANSYS FEA is being directly feed into the objective function value.

Design variables: During design of a Pressure Vessel, several parameters have to be considered to manufacture it efficiently by meeting up the industry requirements. For the analysis of the current pressure vessel equipment, we consider the head shape, nozzle locations and distance between the supports of the vessel as the design parameters. During Optimization, the design parameters are varied in a specified range.

(1) The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material.

(2) Three combinations of nozzle positions are chosen as inlet and outlet. They are: i) the shell's intake and outflow nozzles are placed at the top and side, respectively, 25 percent of the way from one end to the other. ii) The inlet

nozzle is located on the head and the outlet nozzle is situated at a distance of 25 percent of the total length of the shell from one end and iii) Inlet and outlet nozzles are located at the top and side of the shell, respectively, at distances of 50 percent of the total length of the shell from one end, or in the middle of the shell.

(3) The supports value is varied from1m to 2m. The impact of the distance between two selected saddle supports is investigated. The three situations taken into account in this research are as follows: i) 1 m between the support's and shell's centres ii) 1.5 m distance between centre of shell of and centre of the support and iii) a distance of 2 metres separate the centre of the shell from the centre of the support.

5.1.1 Effect of different head shapes

The three predominantly used head shapes are hemispherical, elliptical, torispherical with different thickness according to ASME standards are subjected to an internal pressure of 10MPa to evaluate maximum stress and deformation contours in each. The different figures displayed below, shows the design variables used in this optimization problem. Results and FEM simulation

The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material. Figures 5.1, 5.2, 5.3, .5.4, 5.5.and 5.6 show stress contour and deformation contours and their respective values of horizontal pressure vessel. Fig. 5.1 shows stress contour of Ellipsoidal head and Fig.5.2 shows total deformation contour of ellipsoidal head. Fig. 5.3 shows stress contour of hemispherical head and Fig.5.4 shows total deformation contour of hemispherical head. And Fig. 5.5 shows stress contour of Torispherical head and Fig.5.6 shows total deformation contour of Torispherical head.

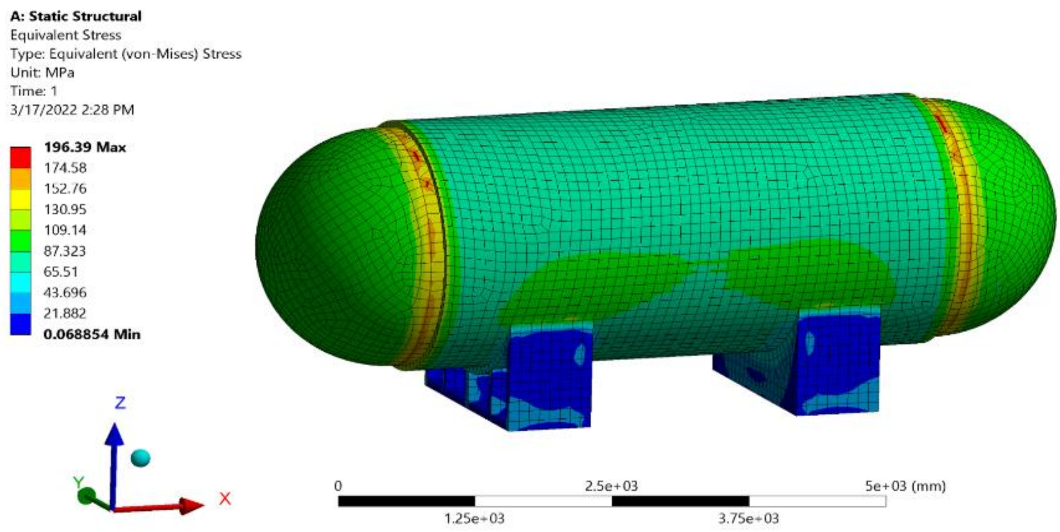


Fig. 5.1: Stress contour in hemispherical head

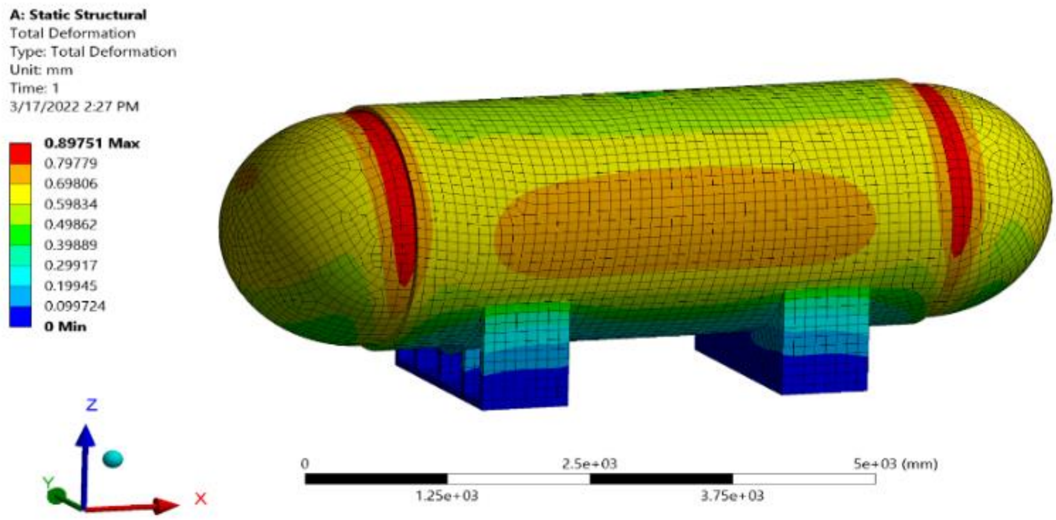


Fig. 5.2: Total deformation contour in hemispherical head

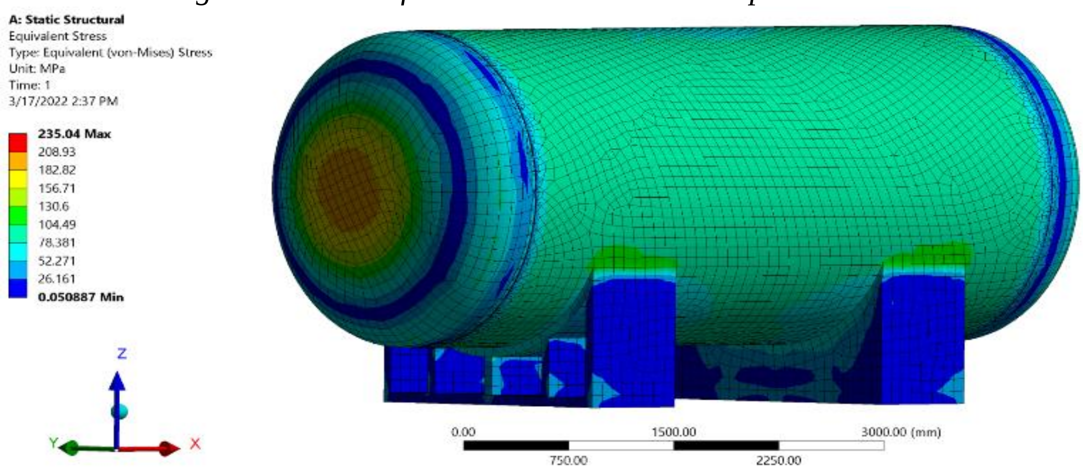


Fig. 5.3: Stress contour in ellipsoidal head

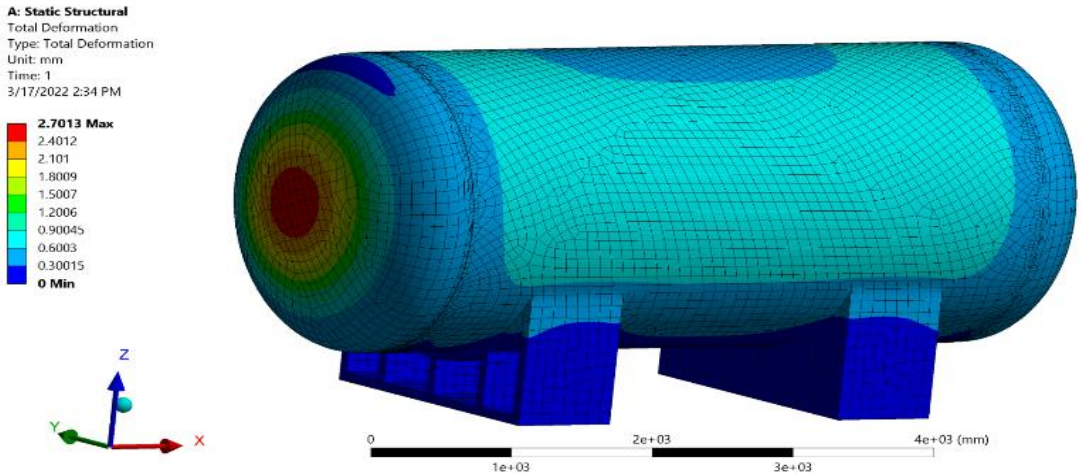


Fig. 5.4: Total deformation contour in ellipsoidal head

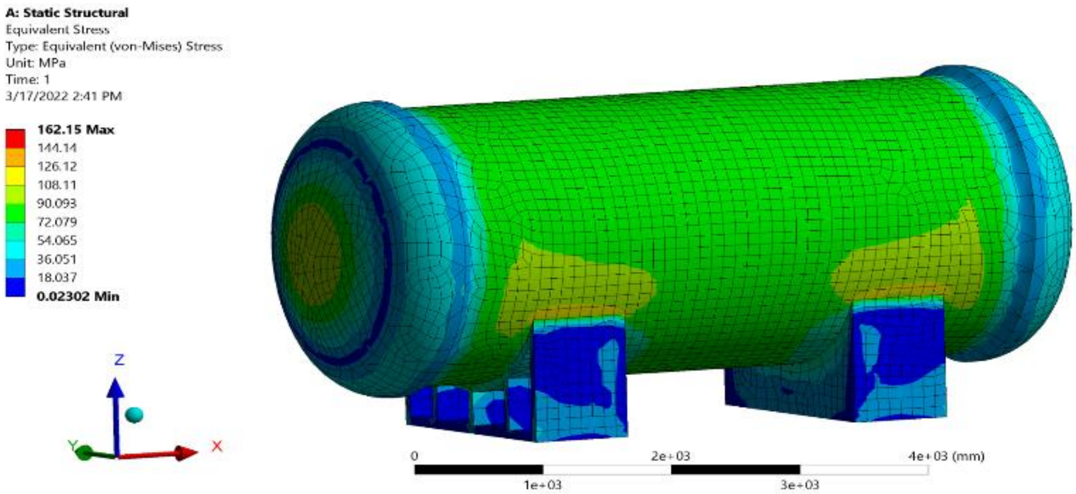


Fig. 5.5: Stress contour in torispherical head

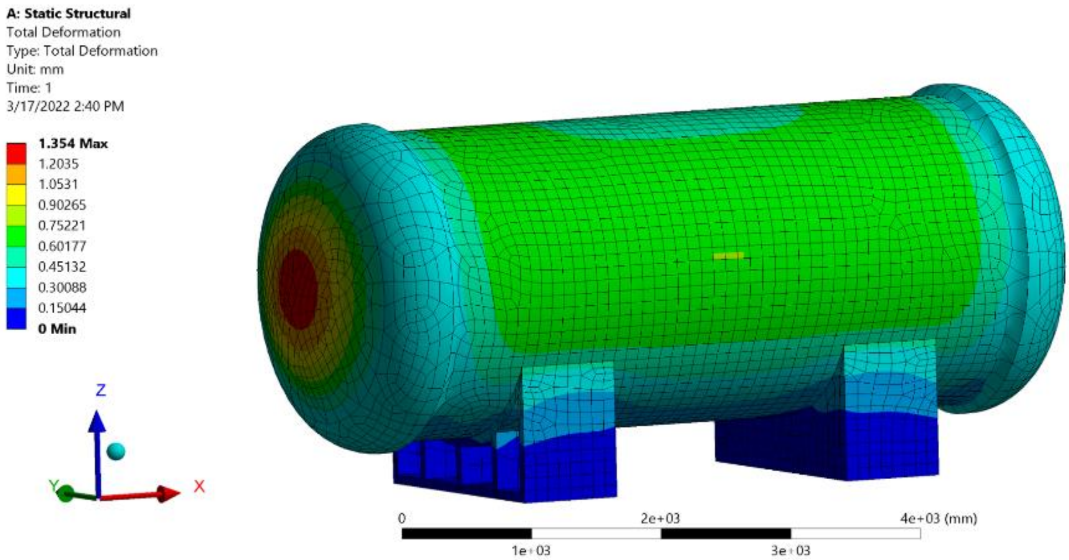


Fig. 5.6: Total deformation contour in torispherical head

It may be observed that from Table 5.1 that torispherical head is much safer for the same material and internal pressure. So, in further analysis, torispherical head horizontal pressure vessel will be used.

Table 5.1: Magnitude of maximum stress and total deformation in different head shapes:

Sl.no.	Head shape	Maximum von-misses stress (MPa)		Maximum Total Deformation (mm)	
		Reference paper	Present	Reference paper	Present
01.	Hemispherical	214.2	196.39	1.1884	0.897
02.	Ellipsoidal	206.94	235.04	2.4625	2.7013
03.	Torispherical	195.08	162.15	1.7642	1.354

5.1.2 Effect of different nozzle positions

Three combinations of nozzle positions are chosen as inlet and outlet. Figures 5.7, 5.8, 5.9, 5.10, 5.11 and 5.12 shows stress and total deformation contour of horizontal pressure vessel having different nozzle positions with torispherical head. Fig. 5.7 and fig.5.8 shows the stress and total deformation contour of inlet and outlet nozzle located at top and side of the shell at a distance of 50% of total length of shell from one end respectively, i.e. at the centre of the shell. As shown in fig 5.9 and fig.5.10 shows inlet and outlet nozzle located at top and side of the shell at a distance of 25% of total length of shell from one end respectively and fig 5.11 and fig.5.12 shows outlet nozzle on the head and inlet nozzle located at a distance of 25% of total length of shell from one end

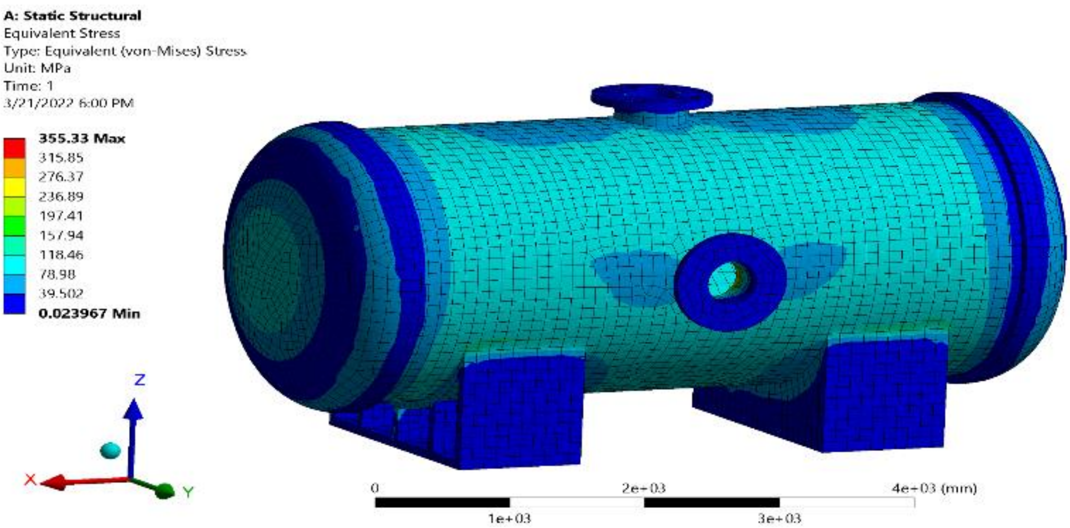


Fig. 5.7: Stress contour for nozzles at centre of the shell

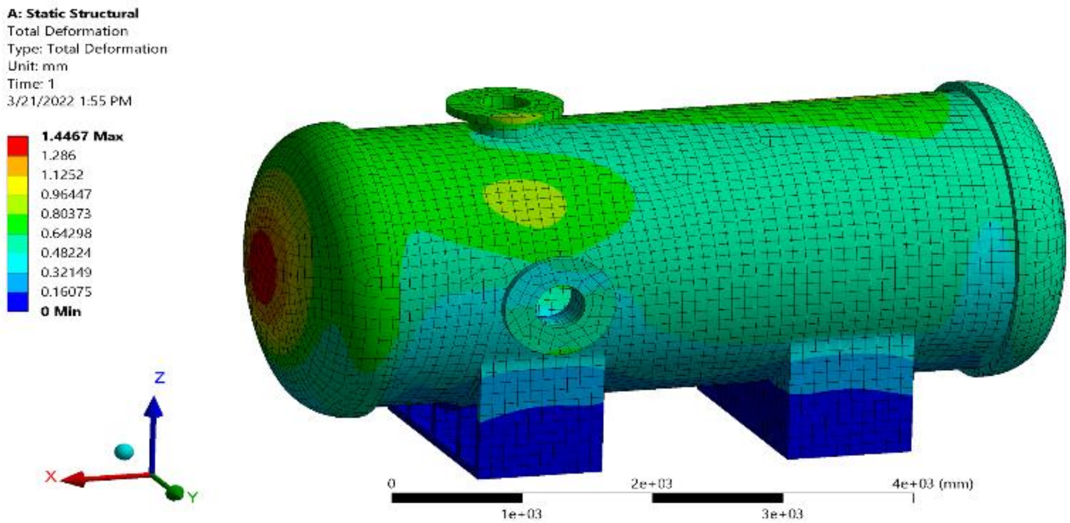


Fig. 5.8: Total deformation contour for nozzles at centre of the shell

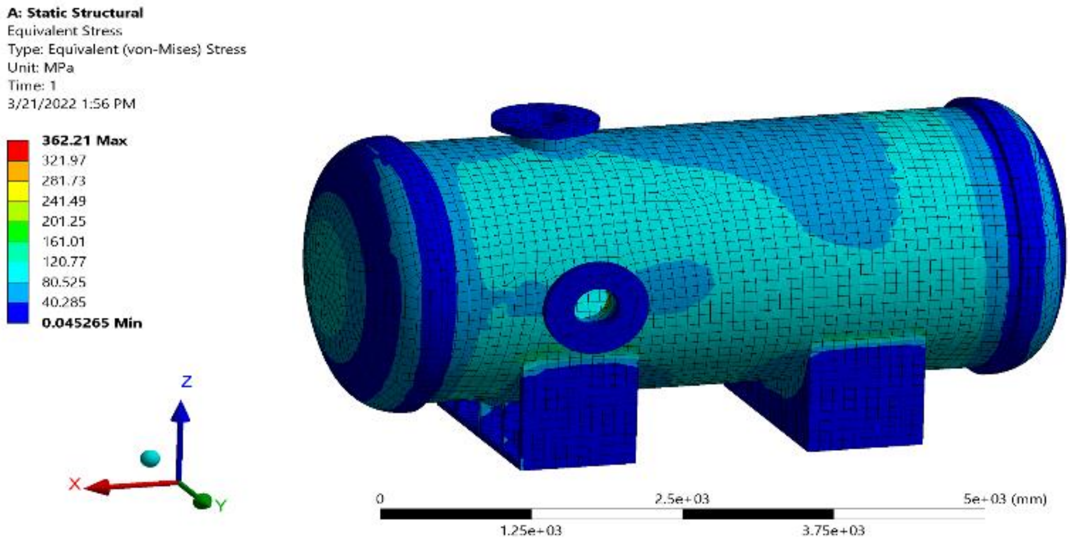


Fig. 5.9: Stress contour for nozzles at $1/4^{th}$ of total length of the shell

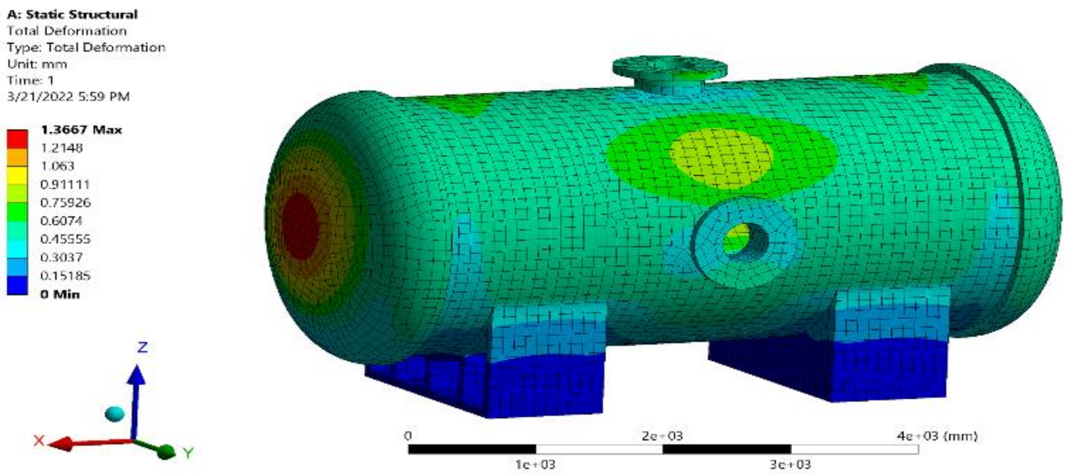


Fig. 5.10: Total deformation contour for nozzles at $1/4^{th}$ of total length of the shell

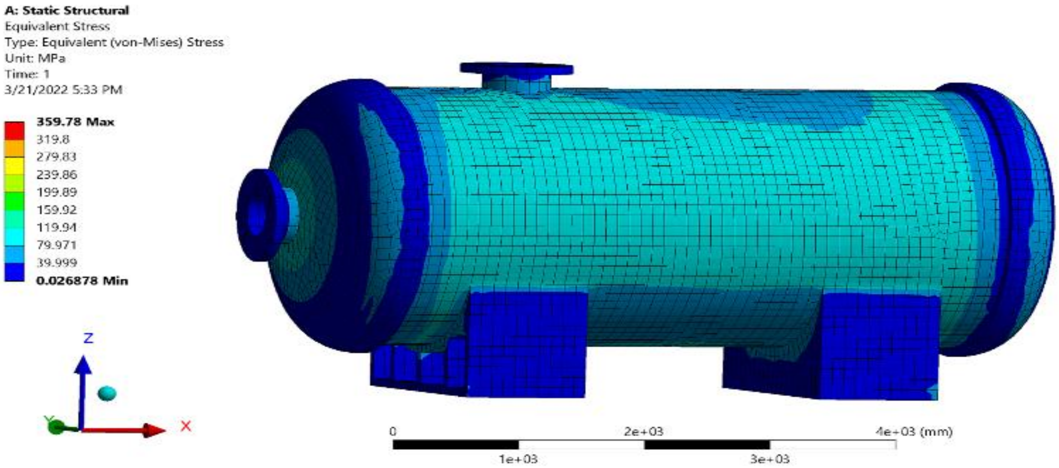


Fig. 5.11: Stress contour for one nozzle at head and one at 1/4th distance of the shell

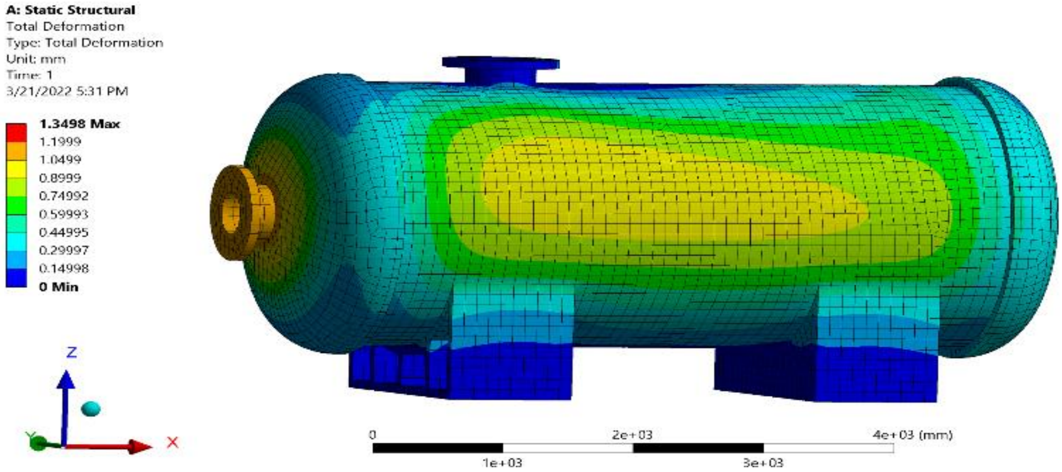


Fig. 5.12: Total deformation contour for one nozzle at head and one at 1/4th distance of the shell

It is observed that the maximum stress induced when the nozzles are located at the centre of the shell is less which is presented in Table 5.2. Hence, the nozzles are located at this position.

Table 5.2: Maximum stress and total deformation for different nozzle positions

Sl.no.	Nozzle position	Maximum von-misses stress (MPa)		Maximum Total Deformation(mm)	
		Reference paper	Present	Reference paper	Present
01.	Nozzle at the centre of the shell	358.66	355.33	3.4593	1.3667
02.	At one fourth of the total length of the shell	378.56	362.21	4.9475	1.4467
03.	The nozzle at the head and one on the shell at one fourth distance	384.65	359.78	1.7718	1.3498

5.1.3 Effect of distance between supports

The supports value is varied from 1m to 2m. Two saddle supports are chosen and the effect of distance between them is examined and stress and deformation contours are shown in figures 5.13, 5.14, 5.15, 5.16, 5.17, and 5.18. The stress and deformation contours are shown in fig.5.13 and fig.5.14 when 1 m distance between centre of shell of and centre of the support. The stress and deformation contours shown in Fig 5.15 and Fig.5.16 when 1.5 m distance between centre of shell of and centre of the support and the stress and deformation contours shown in fig 5.17 and 5.18 when 2 m distance between centre of shell of and centre of the support.

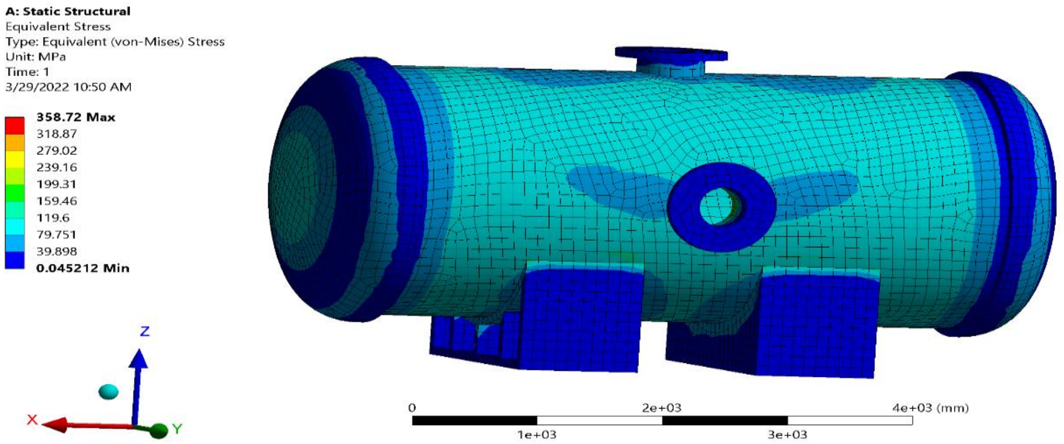


Fig. 5.13: Stress contour for case distance between supports of 1m.

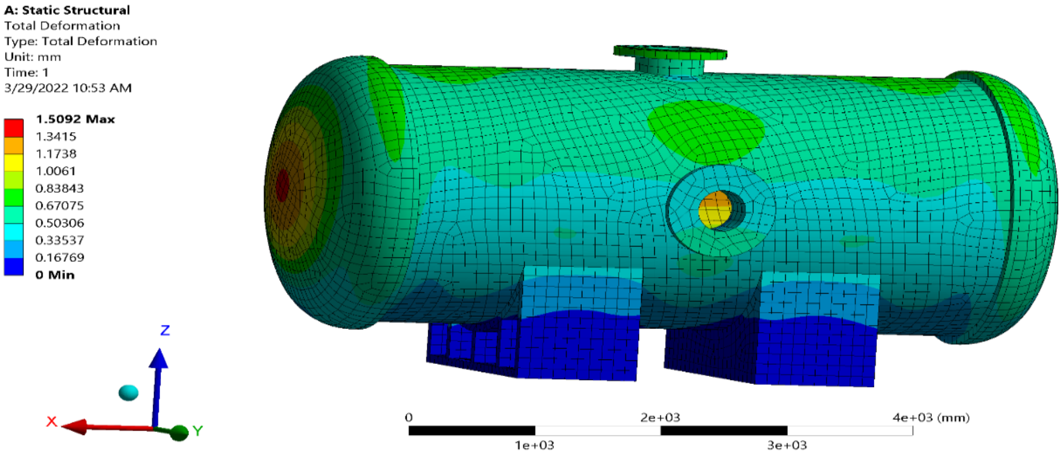


Fig. 5.14: Total deformation contour for case distance between supports of 1m

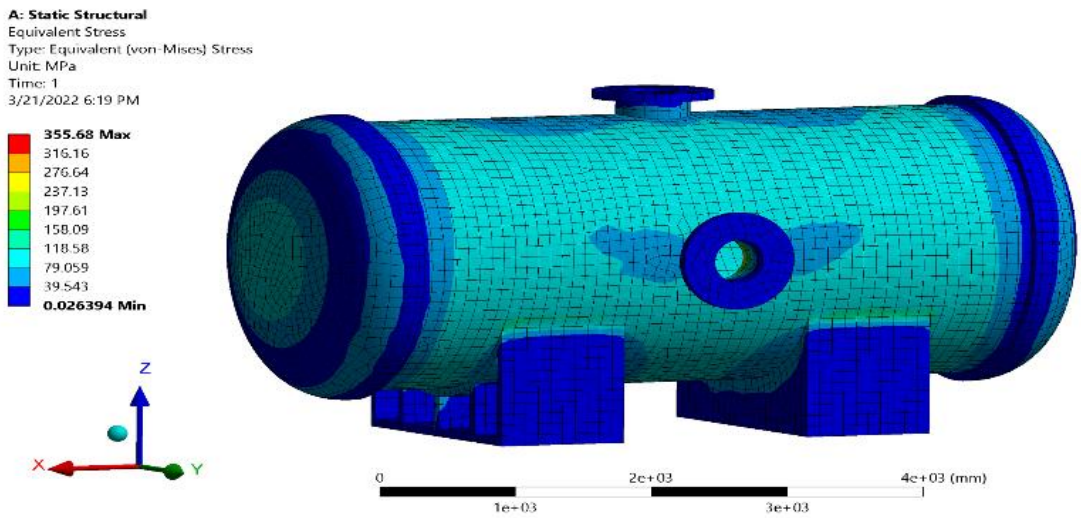


Fig. 5.15: Stress contour for case distance between supports of 1.5m.

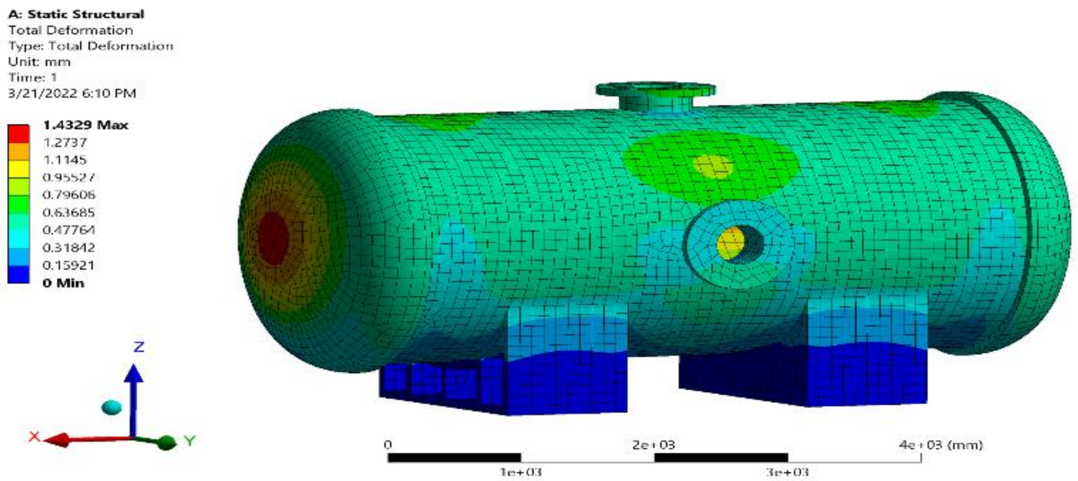


Fig. 5.16: Total deformation contour for case distance between supports of 1.5m.

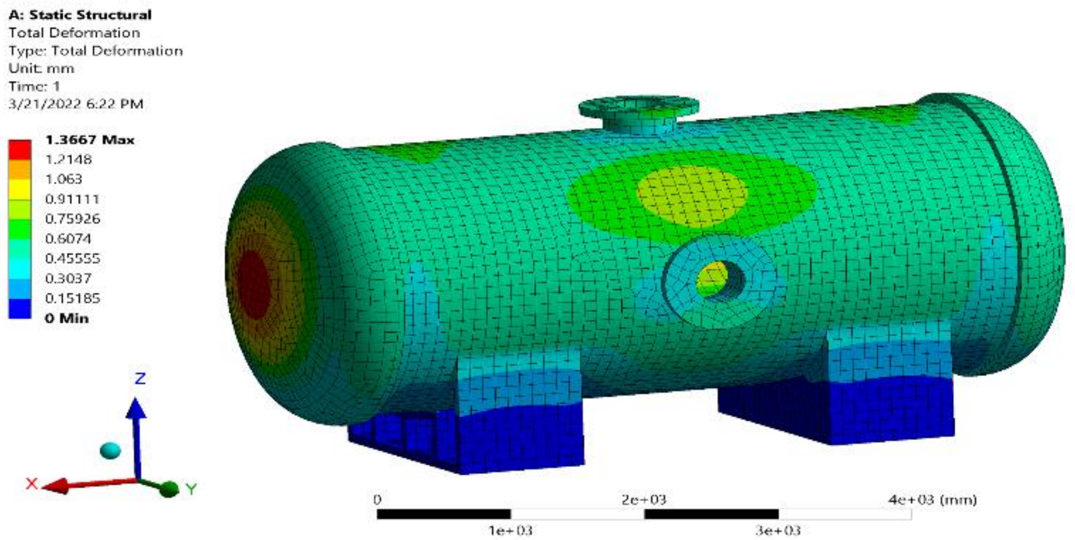


Fig. 5.17: Total deformation contour for case distance between supports of 2m.

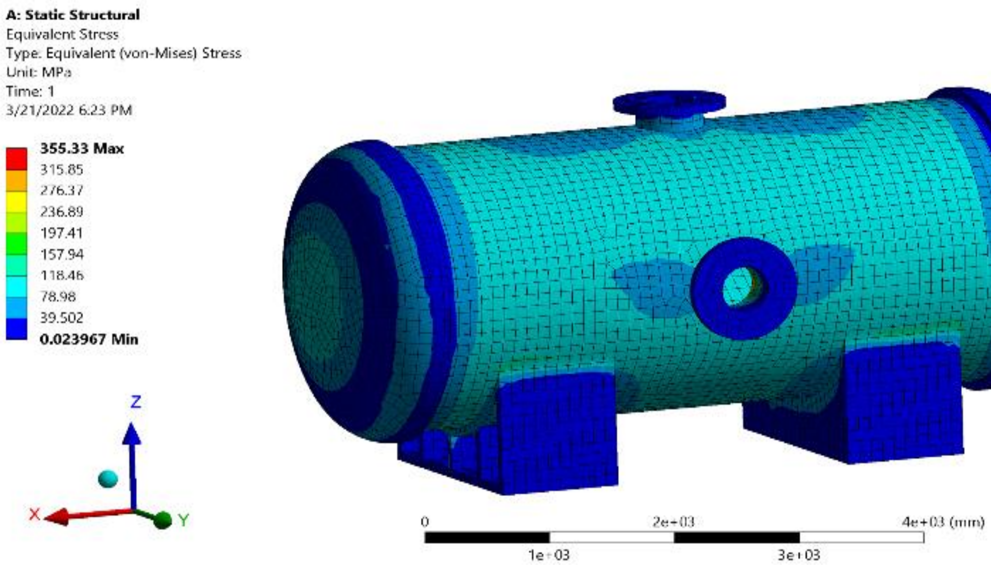


Fig. 5.18: Stress contour for case distance between supports of 2m.

Table 5.3 represents that the least value of stress is generated when the distance between the supports are at 2m. Hence we observed that the least value of stress will be generated when the supports are at extreme ends.

Table 5.3: Maximum stress and total deformation for different support positions.

Sl.no.	**Distance between the supports (m)**	**Maximum von-misses stress (MPa)**		**Maximum Total Deformation (mm)**	
		Reference paper	Present	Reference paper	Present
01.	1	359.44	356.68	4.2085	1.4329
02.	1.5	358.66	356.76	3.4593	1.3668
03.	2	358.37	357.06	2.7497	1.3081

CHAPTER 5: PARAMETRIC OPTIMIZATION

5.1 Optimization problem

Optimization may be described as the overall process of looking for the best design that is currently possible. By controlling a group of variable factors that have an impact on both the objectives and the design constraints, mathematical optimization is the process of maximising and/or decreasing one or more objectives while maintaining the established design constraints. It's crucial to understand that in order to use mathematical optimization, the objective(s) and design constraint(s) must both be expressed as quantitative functions of the variable parameters. Design variables and choice variables are other names for these variable elements.

Objective function: The objective function to be minimized is the maximum von misses stress and total deformation of the pressure vessel. Like most of the conventional optimization problems, this is also a single objective optimization problem. Instead of the conventional method where the objective evaluated as a function of design parameters, a scalar value representing the total volume of the pressure vessel structure, obtained from ANSYS FEA is being directly feed into the objective function value.

Design variables: During design of a Pressure Vessel, several parameters have to be considered to manufacture it efficiently by meeting up the industry requirements. For the analysis of the current pressure vessel equipment, we consider the head shape, nozzle locations and distance between the supports of the vessel as the design parameters. During Optimization, the design parameters are varied in a specified range.

(1) The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material.

(2) Three combinations of nozzle positions are chosen as inlet and outlet. They are: i) the shell's intake and outflow nozzles are placed at the top and side, respectively, 25 percent of the way from one end to the other. ii) The inlet

nozzle is located on the head and the outlet nozzle is situated at a distance of 25 percent of the total length of the shell from one end and iii) Inlet and outlet nozzles are located at the top and side of the shell, respectively, at distances of 50 percent of the total length of the shell from one end, or in the middle of the shell.

(3) The supports value is varied from1m to 2m. The impact of the distance between two selected saddle supports is investigated. The three situations taken into account in this research are as follows: i) 1 m between the support's and shell's centres ii) 1.5 m distance between centre of shell of and centre of the support and iii) a distance of 2 metres separate the centre of the shell from the centre of the support.

5.1.1 Effect of different head shapes

The three predominantly used head shapes are hemispherical, elliptical, torispherical with different thickness according to ASME standards are subjected to an internal pressure of 10MPa to evaluate maximum stress and deformation contours in each. The different figures displayed below, shows the design variables used in this optimization problem. Results and FEM simulation

The three predominantly used head shapes are, hemispherical, elliptical, torispherical are chosen for the same internal pressure and material. Figures 5.1, 5.2, 5.3, .5.4, 5.5.and 5.6 show stress contour and deformation contours and their respective values of horizontal pressure vessel. Fig. 5.1 shows stress contour of Ellipsoidal head and Fig.5.2 shows total deformation contour of ellipsoidal head. Fig. 5.3 shows stress contour of hemispherical head and Fig.5.4 shows total deformation contour of hemispherical head. And Fig. 5.5 shows stress contour of Torispherical head and Fig.5.6 shows total deformation contour of Torispherical head.

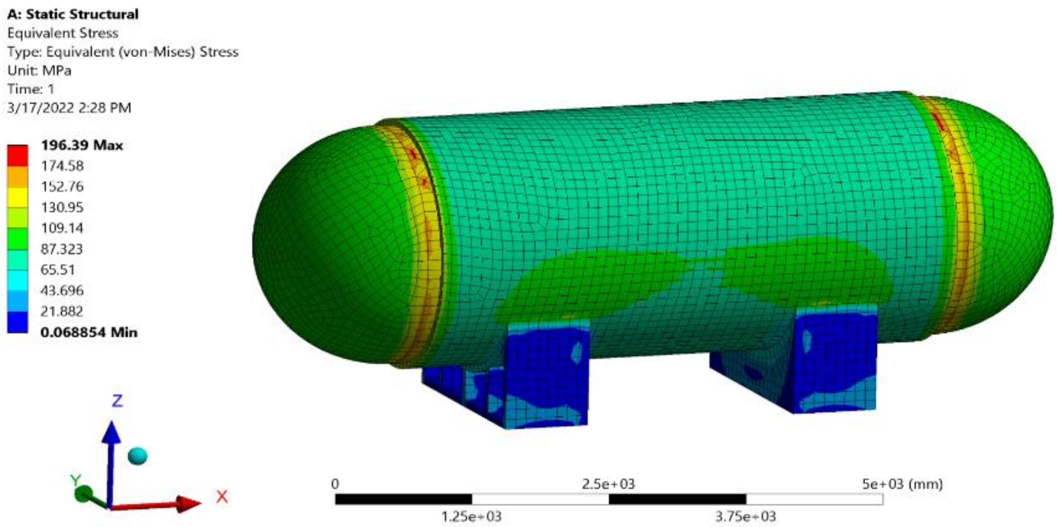


Fig. 5.1: Stress contour in hemispherical head

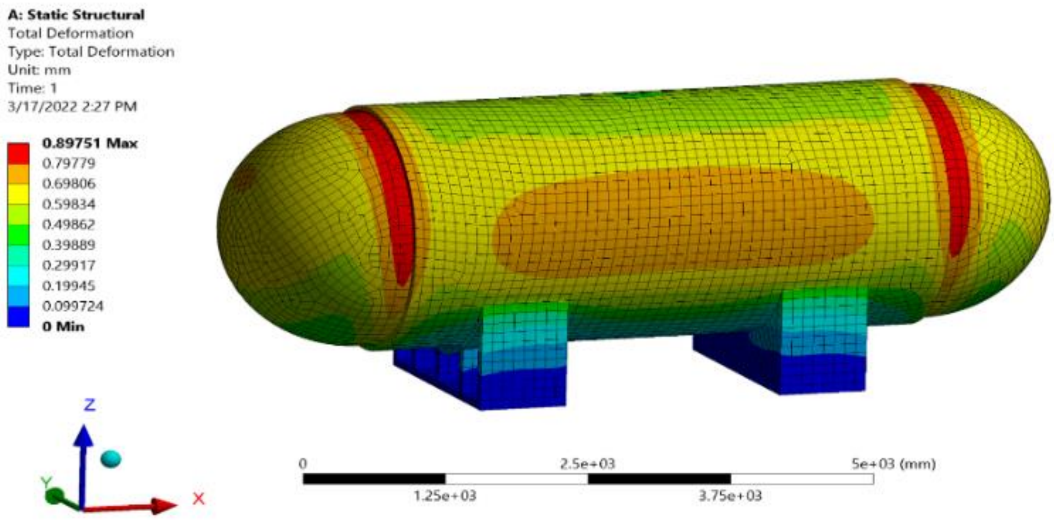


Fig. 5.2: Total deformation contour in hemispherical head

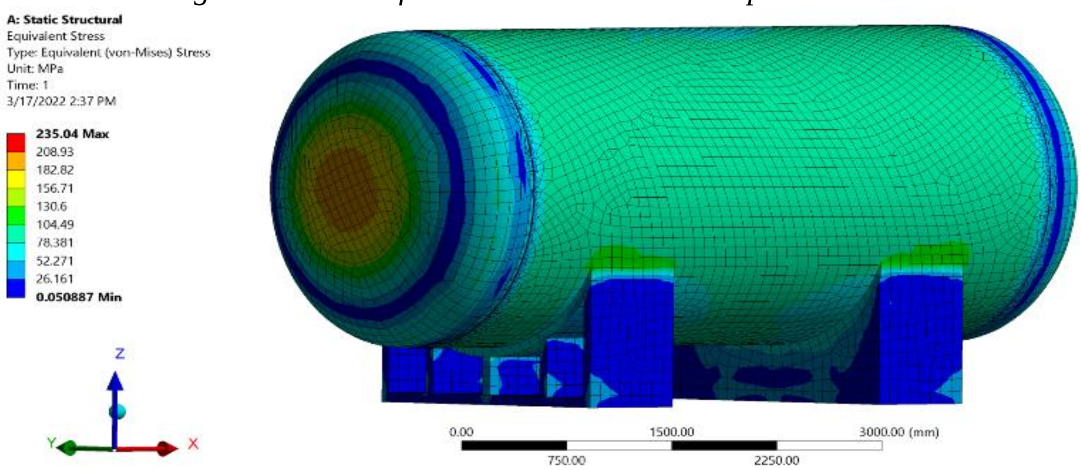


Fig. 5.3: Stress contour in ellipsoidal head

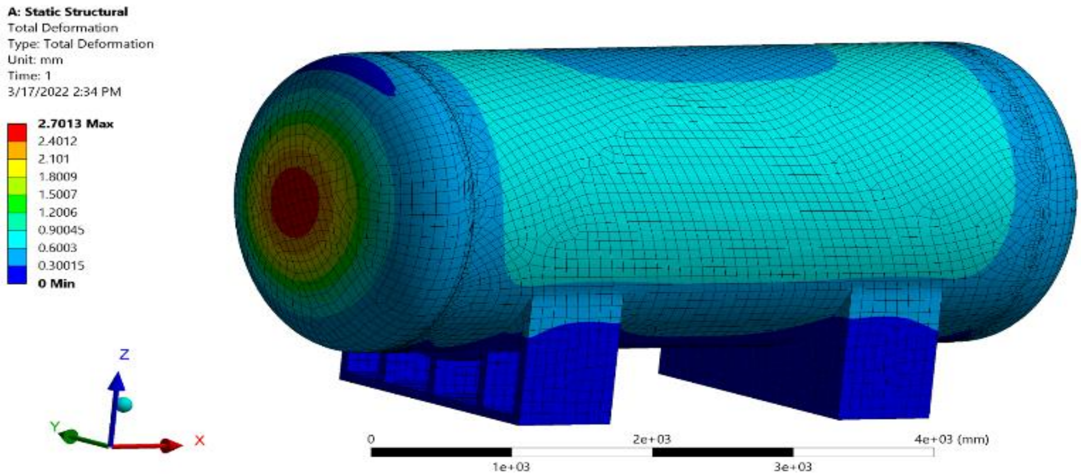


Fig. 5.4: Total deformation contour in ellipsoidal head

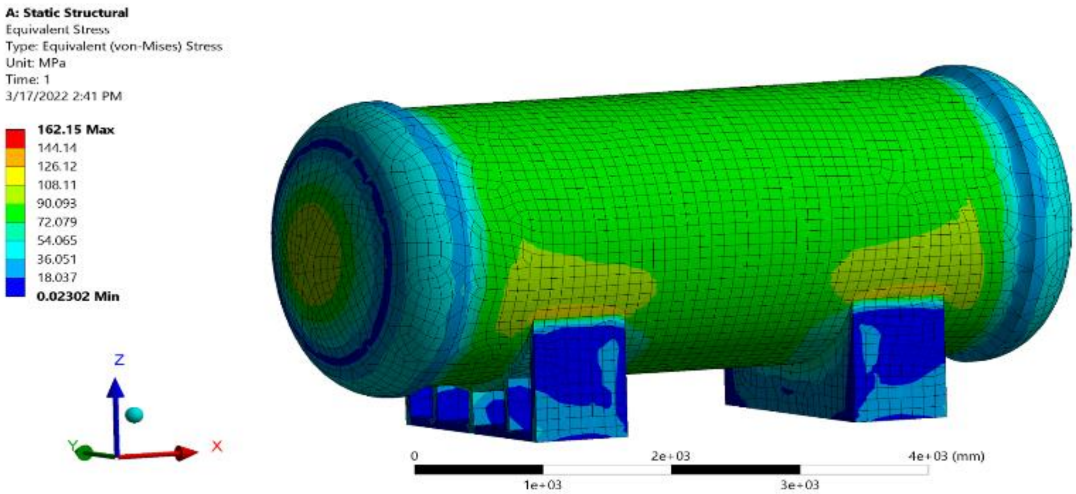


Fig. 5.5: Stress contour in torispherical head

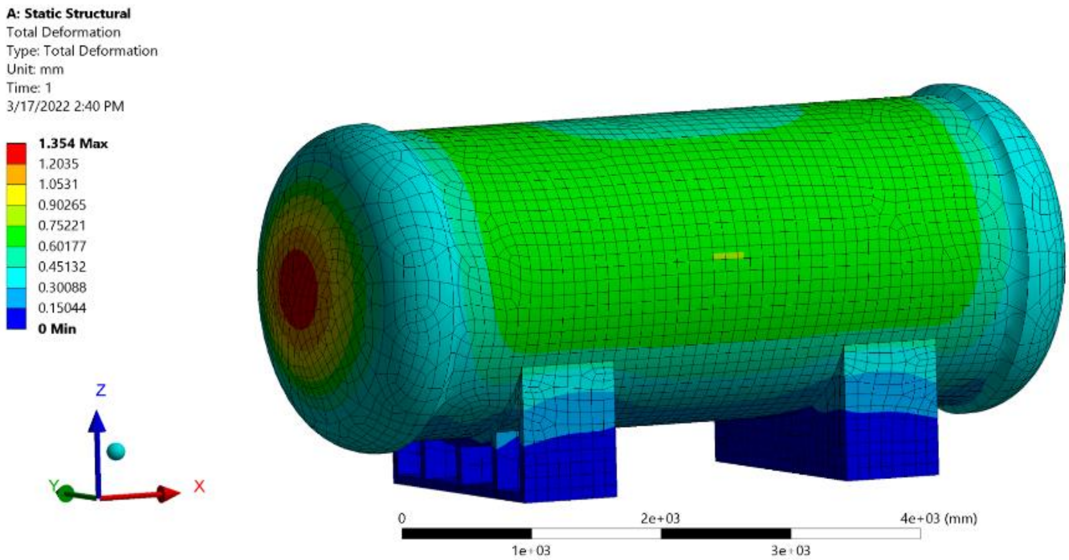


Fig. 5.6: Total deformation contour in torispherical head

It may be observed that from Table 5.1 that torispherical head is much safer for the same material and internal pressure. So, in further analysis, torispherical head horizontal pressure vessel will be used.

Table 5.1: Magnitude of maximum stress and total deformation in different head shapes:

Sl.no.	Head shape	**Maximum von-misses stress (MPa)**		**Maximum Total Deformation (mm)**	
		Reference paper	Present	Reference paper	Present
01.	Hemispherical	214.2	196.39	1.1884	0.897
02.	Ellipsoidal	206.94	235.04	2.4625	2.7013
03.	Torispherical	195.08	162.15	1.7642	1.354

5.1.2 Effect of different nozzle positions

Three combinations of nozzle positions are chosen as inlet and outlet. Figures 5.7, 5.8, 5.9, 5.10, 5.11 and 5.12 shows stress and total deformation contour of horizontal pressure vessel having different nozzle positions with torispherical head. Fig. 5.7 and fig.5.8 shows the stress and total deformation contour of inlet and outlet nozzle located at top and side of the shell at a distance of 50% of total length of shell from one end respectively, i.e. at the centre of the shell. As shown in fig 5.9 and fig.5.10 shows inlet and outlet nozzle located at top and side of the shell at a distance of 25% of total length of shell from one end respectively and fig 5.11 and fig.5.12 shows outlet nozzle on the head and inlet nozzle located at a distance of 25% of total length of shell from one end

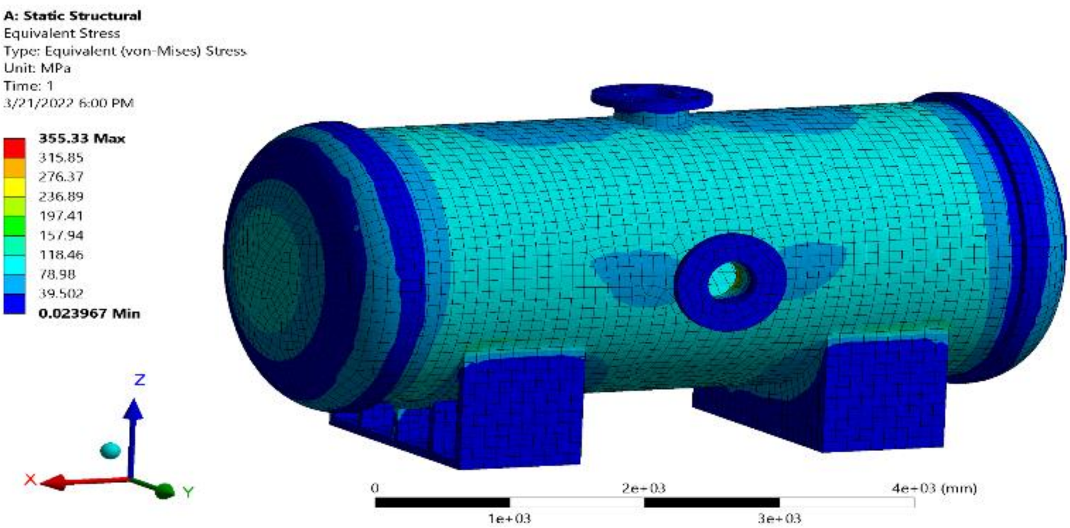


Fig. 5.7: Stress contour for nozzles at centre of the shell

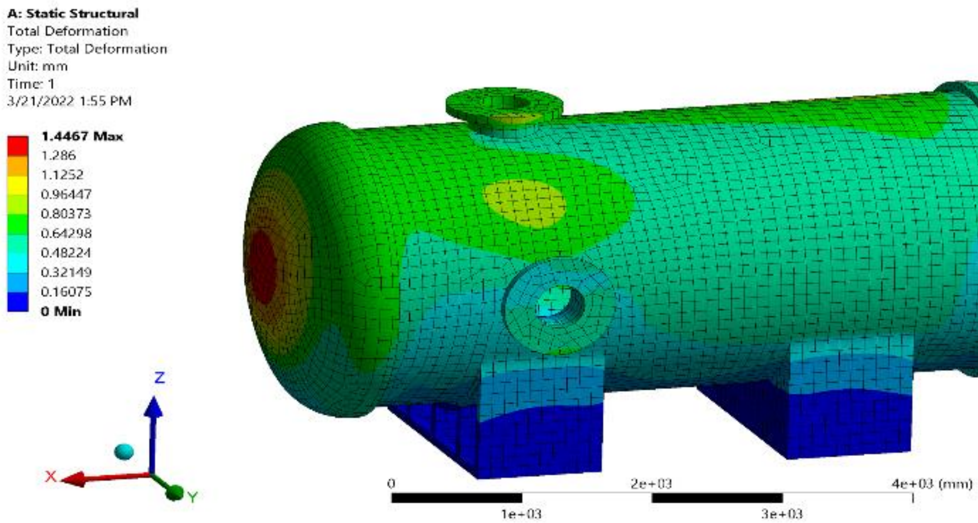


Fig. 5.8: Total deformation contour for nozzles at centre of the shell

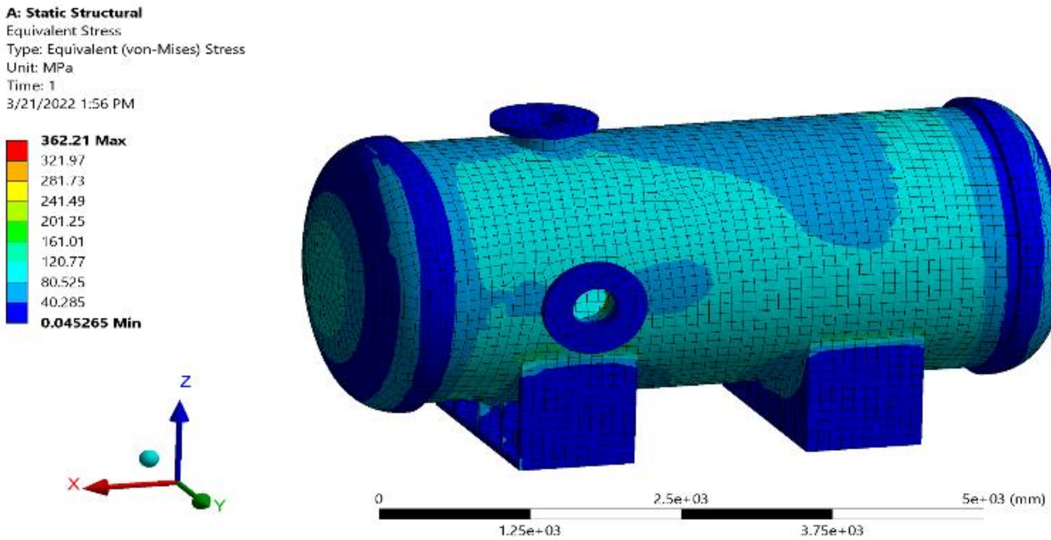


Fig. 5.9: Stress contour for nozzles at 1/4th of total length of the shell

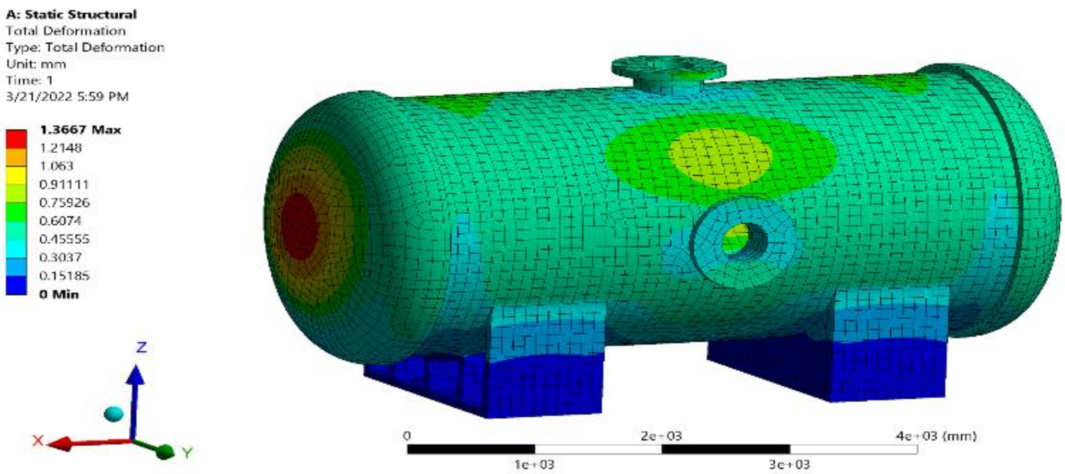


Fig. 5.10: Total deformation contour for nozzles at 1/4th of total length of the shell

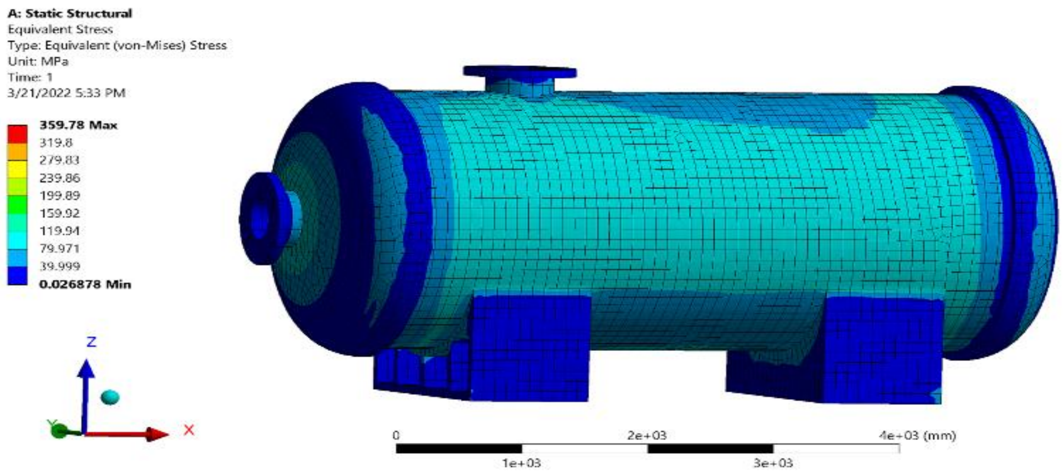


Fig. 5.11: Stress contour for one nozzle at head and one at 1/4th distance of the shell

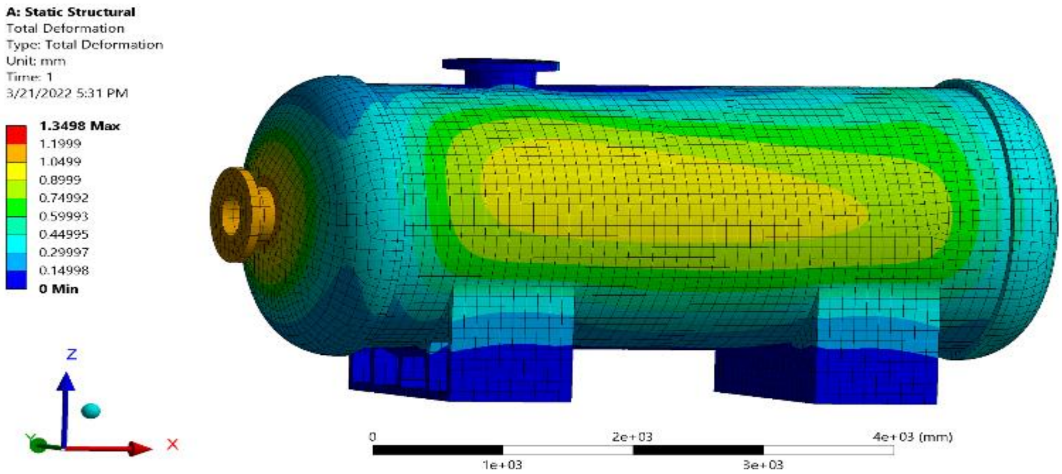


Fig. 5.12: Total deformation contour for one nozzle at head and one at 1/4th distance of the shell

It is observed that the maximum stress induced when the nozzles are located at the centre of the shell is less which is presented in Table 5.2. Hence, the nozzles are located at this position.

Table 5.2: Maximum stress and total deformation for different nozzle positions

Sl.no.	Nozzle position	Maximum von-misses stress (MPa)		Maximum Total Deformation(mm)	
		Reference paper	Present	Reference paper	Present
01.	Nozzle at the centre of the shell	358.66	355.33	3.4593	1.3667
02.	At one fourth of the total length of the shell	378.56	362.21	4.9475	1.4467
03.	The nozzle at the head and one on the shell at one fourth distance	384.65	359.78	1.7718	1.3498

5.1.3 Effect of distance between supports

The supports value is varied from 1m to 2m. Two saddle supports are chosen and the effect of distance between them is examined and stress and deformation contours are shown in figures 5.13, 5.14, 5.15, 5.16, 5.17, and 5.18. The stress and deformation contours are shown in fig.5.13 and fig.5.14 when 1 m distance between centre of shell of and centre of the support. The stress and deformation contours shown in Fig 5.15 and Fig.5.16 when 1.5 m distance between centre of shell of and centre of the support and the stress and deformation contours shown in fig 5.17 and 5.18 when 2 m distance between centre of shell of and centre of the support.

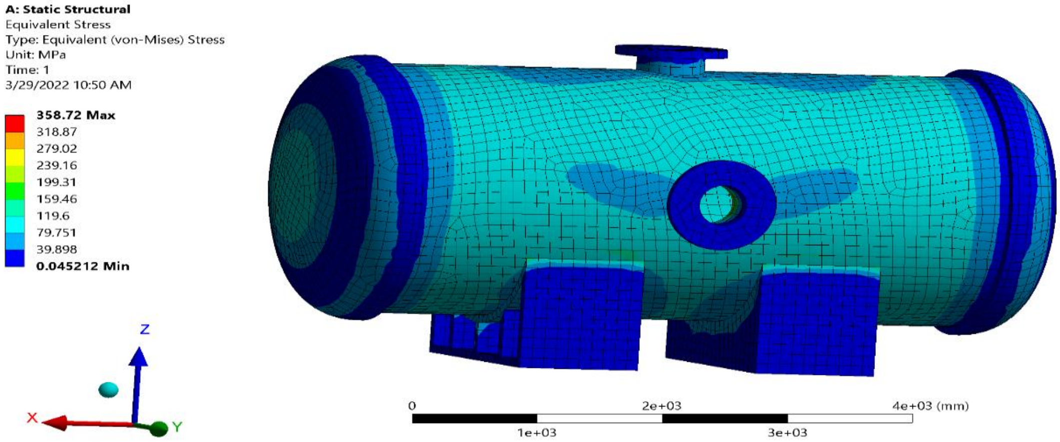


Fig. 5.13: Stress contour for case distance between supports of 1m.

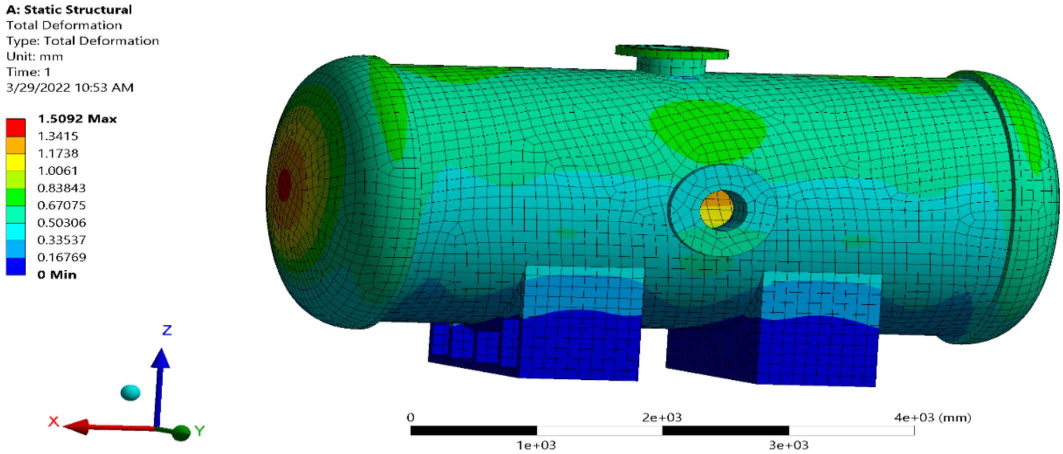


Fig. 5.14: Total deformation contour for case distance between supports of 1m

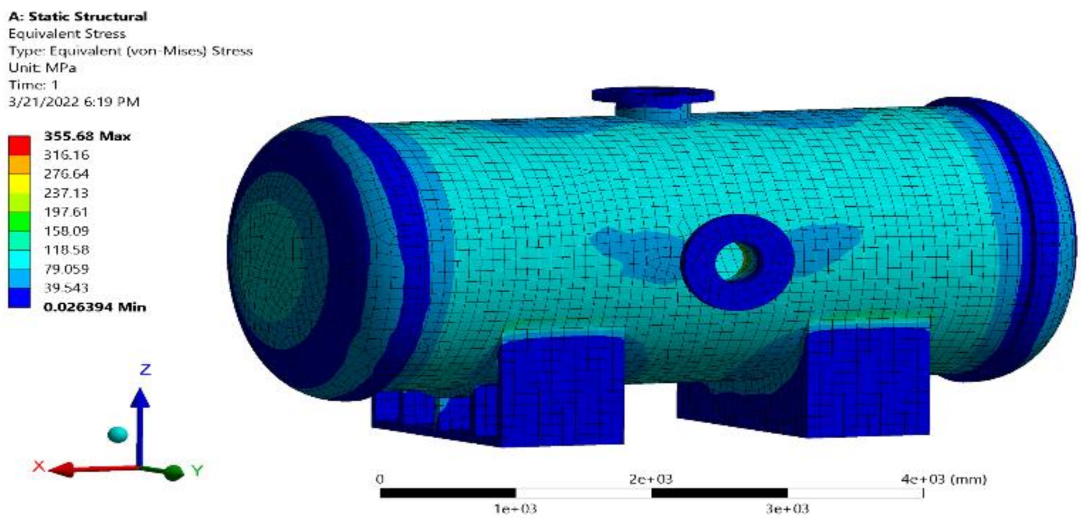


Fig. 5.15: Stress contour for case distance between supports of 1.5m.

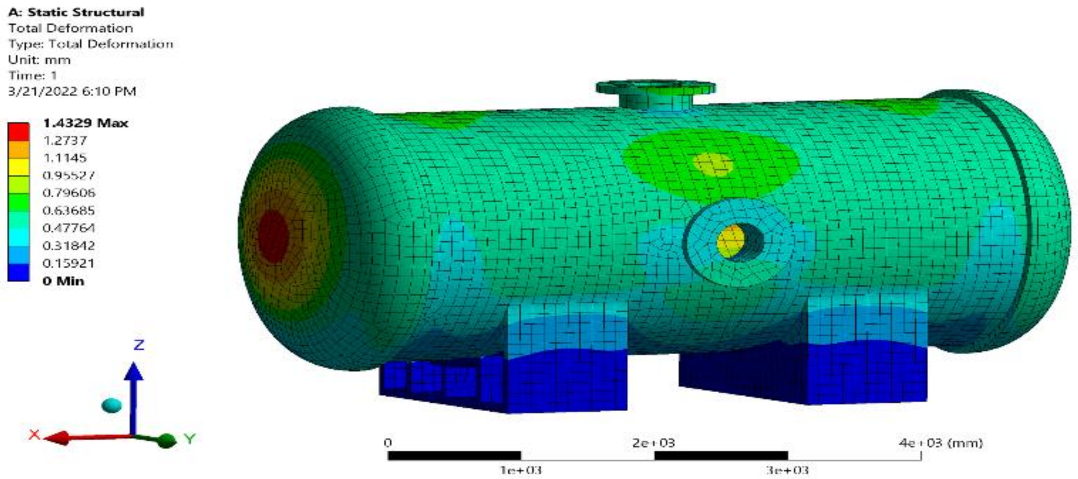


Fig. 5.16: Total deformation contour for case distance between supports of 1.5m.

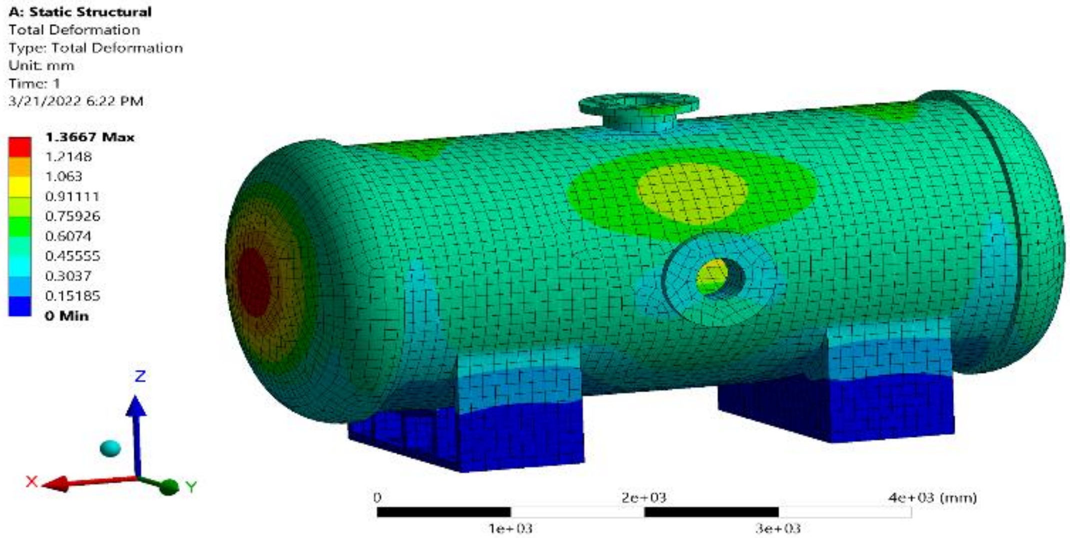


Fig. 5.17: Total deformation contour for case distance between supports of 2m.

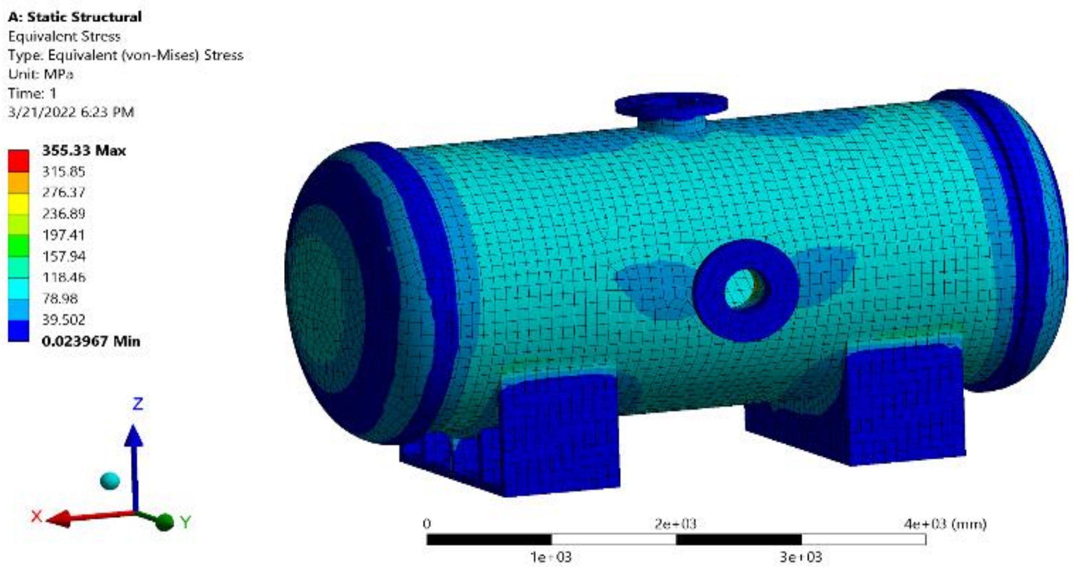


Fig. 5.18: Stress contour for case distance between supports of 2m.

Table 5.3 represents that the least value of stress is generated when the distance between the supports are at 2m. Hence we observed that the least value of stress will be generated when the supports are at extreme ends.

Table 5.3: Maximum stress and total deformation for different support positions.

Sl.no.	**Distance between the supports (m)**	**Maximum von-misses stress (MPa)**		**Maximum Total Deformation (mm)**	
		Reference paper	Present	Reference paper	Present
01.	1	359.44	356.68	4.2085	1.4329
02.	1.5	358.66	356.76	3.4593	1.3668
03.	2	358.37	357.06	2.7497	1.3081

CHAPTER 6: THERMOMECHANICAL STRESS AND DEFORMATIONS

6.1 Results

The design and optimization of horizontal pressure vessel is carried out in this work. The results from the analysis are such as the use of torispherical head for the same materials and internal pressure results in less stress among others also locating nozzles at the centre of the shell improve the stress generation and total deformation. Further the analysis being carried out with supports sufficiently farther from the centre of the shell is the optimized conditions for the design of horizontal pressure.

Effect on the stress due to increasing internal pressure of the vessel

Further analysis of the horizontal pressure vessel have been carried out over internal pressures of 12MPa and 15MPa respectively. Their results are shown in tabular form below. Table 6.1 shows the von-mises stress and total deformation for different head shapes at different pressures.

From the data depicted in table 6.1, in comparison to the three predominantly used head shapes the torispherical head is safer i.e. less stress is developed for the pressures of 12MPa and 15MPa for the same material and internal pressure.

Table 6.1: Effect of different head shapes at 12MPa & 15MPa

Sl. no.	Head shape	Maximum von-mises stress (MPa)				Maximum Total deformation (mm)			
		Reference paper	Pressure at			Reference paper	Pressure at		
			10 MPa	12 MPa	15 MPa		10 MPa	12 MPa	15 MPa
01.	Hemispherical	214.2	196.39	226.45	294.59	1.1884	0.897	1.078	1.3463
02.	Ellipsoidal	206.94	229	280.13	352.56	2.4625	2.7	3.2418	4.052
03.	Torispherical	195.08	167.83	203.41	243.22	1.7642	1.3472	1.6258	2.031

In case of different nozzle positions, up to 12MPa the vessel is safe i.e. within the failure stress of 493Mpa which is also an ultimate tensile strength of the vessel material beyond that the value exceeds. So, among the all nozzle positions, the nozzle at the centre results the least value of stress as shown in table 6.2.

Table 6.2: Effect of different nozzle positions at 12 MPa & 15 MPa

Sl. no	Nozzles position	Maximum von-mises stress (MPa)				Maximum Total deformation (mm)			
		Reference paper	Pressure at			Reference paper	Pressure at		
			10 MPa	12 MPa	15 MPa		10MPa	12MPa	15MPa
01	Nozzle at one head and the other at 1/4th distance of the shell	384.65	359.78	431.73	539.66	1.7718	1.3498	1.6198	2.0248
02	Both at 1/4th of the total length of the shell	378.56	362.21	434.65	543.31	4.9475	1.4467	1.736	2.1701
03	Nozzles at centre of the shell	358.66	355.33	426.39	532.99	3.4593	1.3667	1.64	2.05

Further the analysis have been done through distance between supports and the internal pressures of 12MPa and 15MPa. In case of 15MPa the value of maximum von-mises stress exceeds the ultimate tensile strength of the vessel material, so, the vessel with distance between supports 2m and internal pressure up to 12MPa results best in pressure analysis as presented in the table 6.3.

Table 6.3: Effect of distance between supports at 12 MPa & 15 MPa

Sl. no	Distance between supports (m)	Maximum von-mises stress (MPa)				Maximum Total deformation (mm)			
		Reference paper	Pressure at			Reference paper	Pressure at		
			10MPa	12MPa	15MPa		10MPa	12MPa	15MPa
01	1	359.4	384.69	461.63	538.09	4.2085	1.5092	1.805	2.2638
02	1.5	358.66	355.68	426.81	535.5	3.4593	1.4329	1.7195	2.1494
03	2	358.37	355.33	426.39	532.39	2.7497	1.3667	1.64	2.05

Effect on the stress due to heating of the vessel

As the vessel is used to hold fluids at high temperature and pressure so the studies have been carried out over internal temperature (℃) of 100, 110 and 120 at the inside shell and outside of the shell is exposed to an ambient temperature of 22℃.The results of maximum Von-mises stress and maximum total deformation are shown in tabular form in the table 6.4 for the different categories.

Table 6.4: Effect of different head shapes at internal temperature of 100℃, 110℃ and 120℃

Sl. no.	Head shape	Maximum von-mises stress (MPa)					Maximum Total deformation (mm)				
		Reference paper	Present	Temperature at (℃)			Reference paper	Present	Temperature at (℃)		
				100	110	120			100	110	120
01.	Hemispherical	214.2	196.39	539.53	598.2	661.87	1.1884	0.897	4.260	4.717	5.175
02.	Ellipsoidal	206.94	229	529.32	592.56	455.81	2.4625	2.7	5.558	5.933	6.309
03.	Torispherical	195.08	167.83	529.95	592.6	655.76	1.7642	1.3472	4.263	4.048	5.032

Here the data shown in table 6.4 depicts that the torispherical head with internal temperature of 100℃ is feasible than all other heads in case of stress and total deformation.

Von-mises stress and maximum total deformation are shown in tabular form in the table 6.5 for the different nozzle positions. For the nozzle position at centre results the less total deformation compare to all other positions.

Table 6.5: Effect of different nozzle positions at internal temperature of 100℃, 110℃ and 120℃

Sl.no.	Nozzles position	Maximum von-mises stress (MPa)					Maximum Total deformation (mm)				
		Reference paper	Present	Temperature at (℃)			Reference paper	Present	Temperature at (℃)		
				100	110	120			100	110	120
01.	Nozzle at one head and the other at 1/4th distance of the shell	384.65	359	575	643.3	710.77	1.7718	1.34	4.4354	4.8544	5.27
02.	Both at 1/4th of the total length of the shell	378.56	362	537	600.0	662.86	4.9475	1.44	4.2963	4.6772	5.06
03.	Nozzles at centre of the shell	358.66	355	573	641.3	708.86	3.4593	1.36	4.198	4.5755	4.95

So, from the data shown in table 6.6, the internal temperature of 100℃ and internal pressure of 10MPa is almost same or less than the ultimate tensile strength of the vessel for the distance between supports 2m and total deformation is also least rather than all other cases .So, this indicates that the vessel is safe.

Table 6.6: Effect of distance between supports at internal temperature of 100℃, 110℃ and 120℃

Sl. no.	Distance between supports (m)	Maximum von-mises stress (MPa)					Maximum Total deformation (mm)				
		Reference paper	Present	Temperature at (℃)			Reference paper	Present	Temperature at (℃)		
				100	110	120			100	110	120
01.	1	359.4	384.69	500.65	559.7	618.77	4.2085	1.5092	4.3144	4.7347	5.2163
02.	1.5	358.66	355.68	538.57	602	665.43	3.4593	1.4329	4.2721	4.654	5.0393
03.	2	358.37	355.33	573.71	641.16	708.16	2.7497	1.3667	4.1978	4.5759	4.9545

6.2 Parametric variations

Results have been discussed through the graphical approach.

Graphs have been plotted over internal pressure of 10MPa.The results of maximum Von-mises stress and maximum total deformation are shown in graph form for the different categories.

1. Stress and deformation in head shapes like hemispherical, ellipsoidal and torispherical shown through various graphs plotted below in fig.6.1 and fig.6.2.

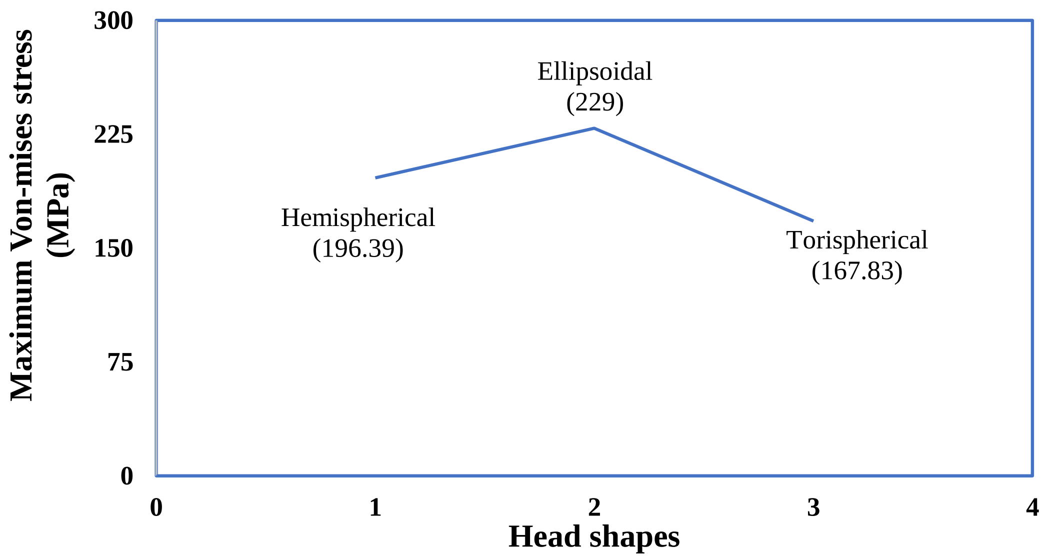


Fig. 6.1: Maximum von-mises stress v/s different head shapes

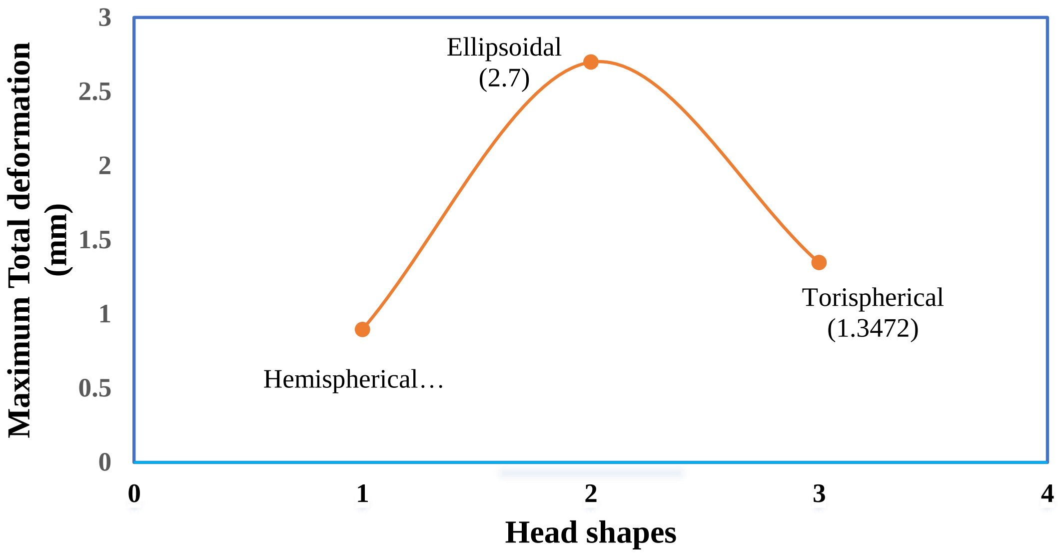


Fig. 6.2: Total deformation v/s different head shapes

It may be observed that from graphs (a) and (b) that torispherical head is much safer for the same material and internal pressure because of the least value. So, in further analysis, torispherical head horizontal pressure vessel will be used.

2. Stress and deformation in different nozzle positions (a) one nozzle at head and one nozzle at 1/4th distance of the shell (b)both the nozzle at 1/4th of total length of the shell (c) both nozzles at the centre of the shell are shown through various graphs plotted below in fig.6.3 and fig.6.4

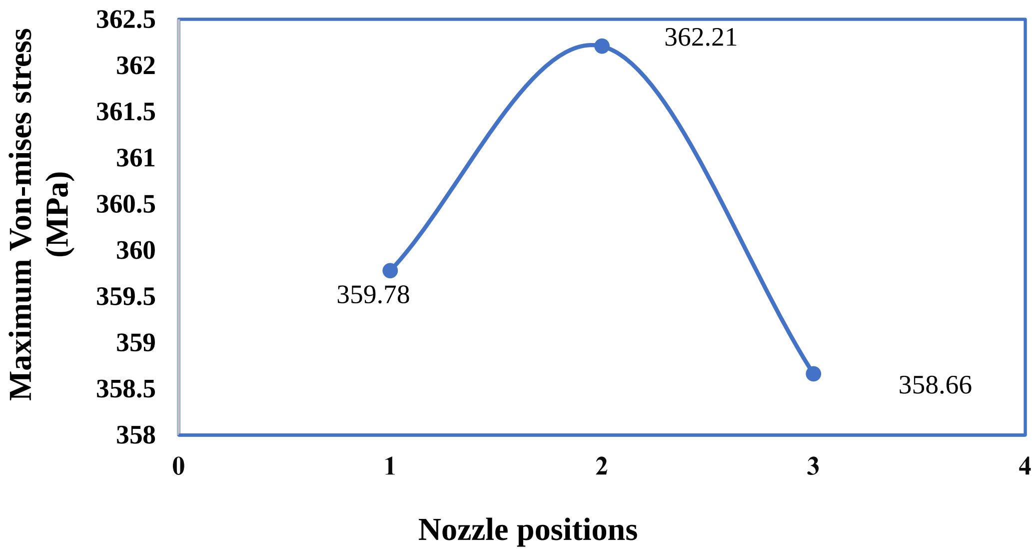


Fig. 6.3: Maximum von-mises stress v/s different nozzle positions

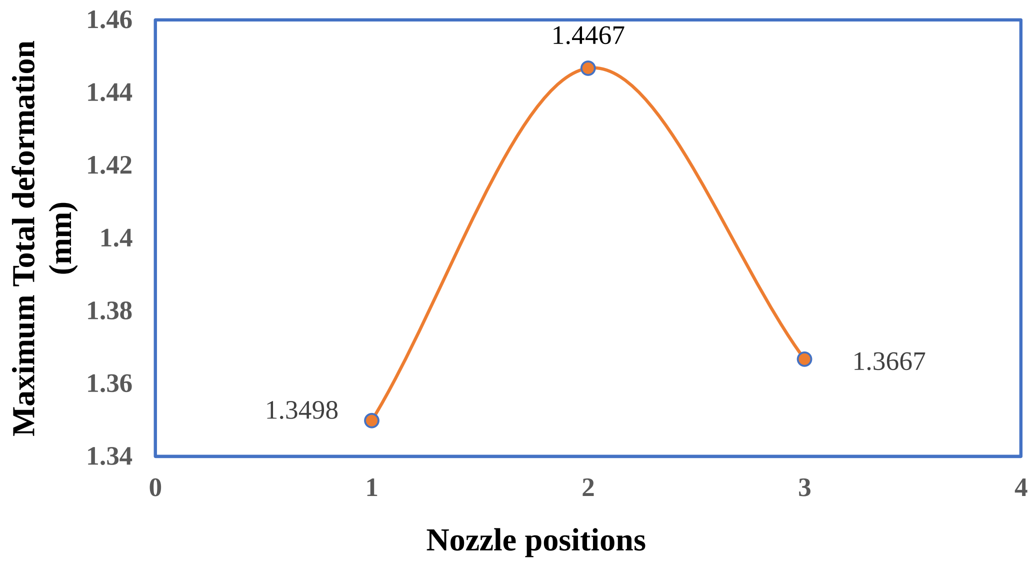


Fig. 6.4: Total deformation v/s different nozzle positions

Minimum of two nozzles are selected, one for inlet and other for outlet for different nozzle positions in horizontal pressure vessel. It may be observed from above graphs (c) and (d) that the maximum stress induced when the nozzles are located at the centre of the shell is less. Hence, the nozzles are located at this position.

3. Stress and deformation in different support positions (a) at 1m (b) at 1.5 m and (c) at 2 m shown through various graphs plotted below in fig.6.5 and fig.6.6.

   Here the pressure vessels are available as beam for which two saddle supports are ideal. Distance between the saddles support is examined by three cases.

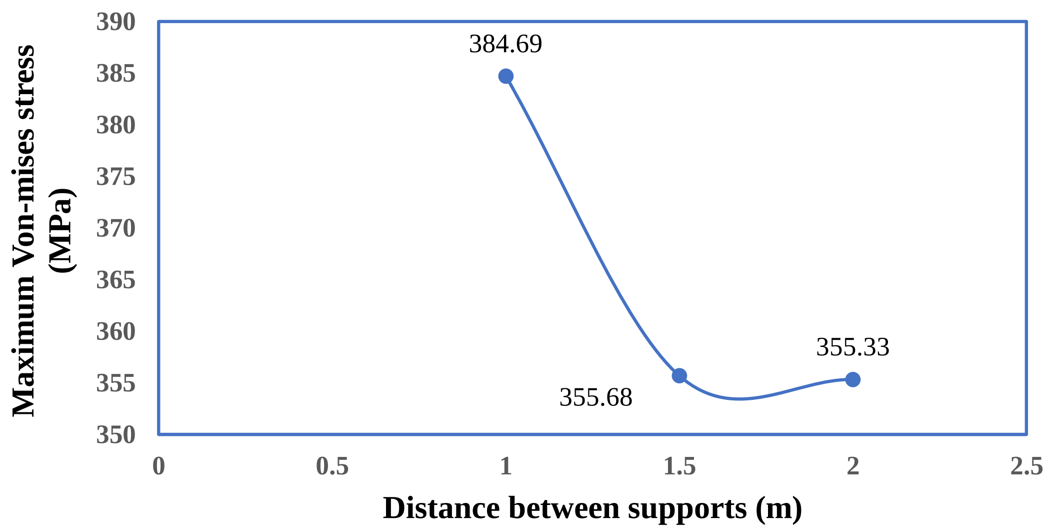


Fig. 6.5: Maximum von-mises stress v/s distance between supports

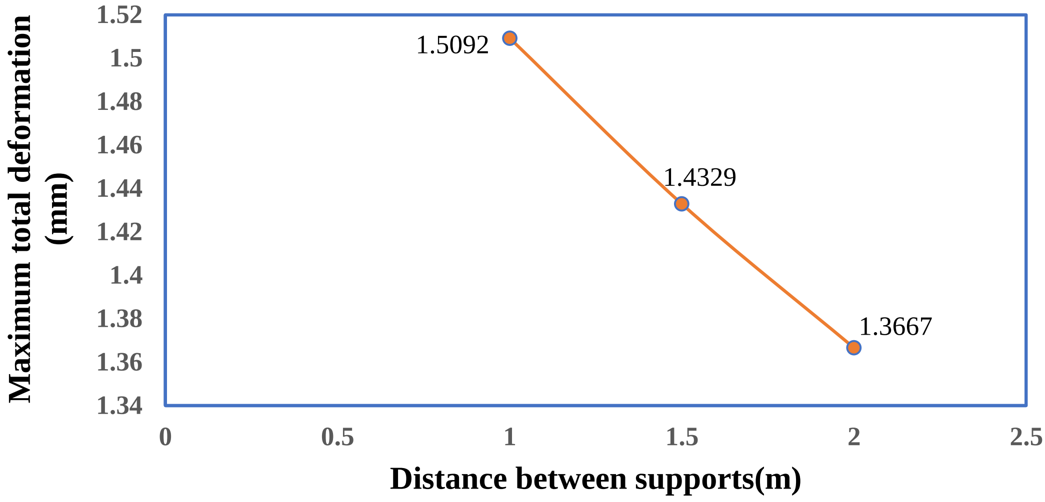


Fig. 6.6: Total deformation v/s distance between supports

It may be seen from graphs drawn in Fig.6.5 and Fig.6.6 that the least value of stress is generated when the distance between the supports are at 2m. Hence, we observed that the least value of stress will be generated when the supports are at extreme ends.

Now, Graphs have been plotted over internal temperature (°C) of 100 at the inside shell and outside of the shell is exposed to an ambient temperature of 22°C.The results of maximum Von-mises stress and maximum total deformation are shown in graph form for the different categories.

4. Stress and deformation in head shapes like hemispherical, ellipsoidal and torispherical shown through various graphs plotted below in fig.6.7 and fig.6.8.

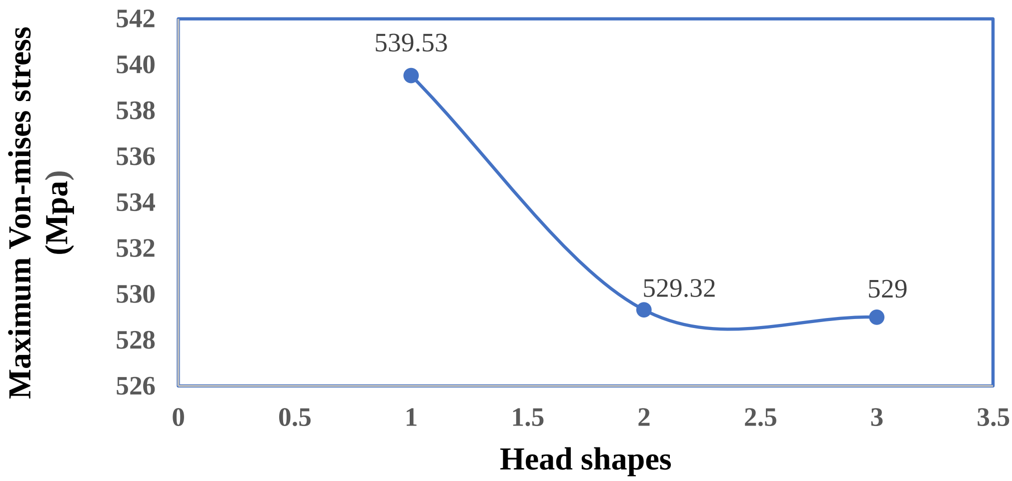


Fig. 6.7: Maximum von-mises stress v/s different head shapes

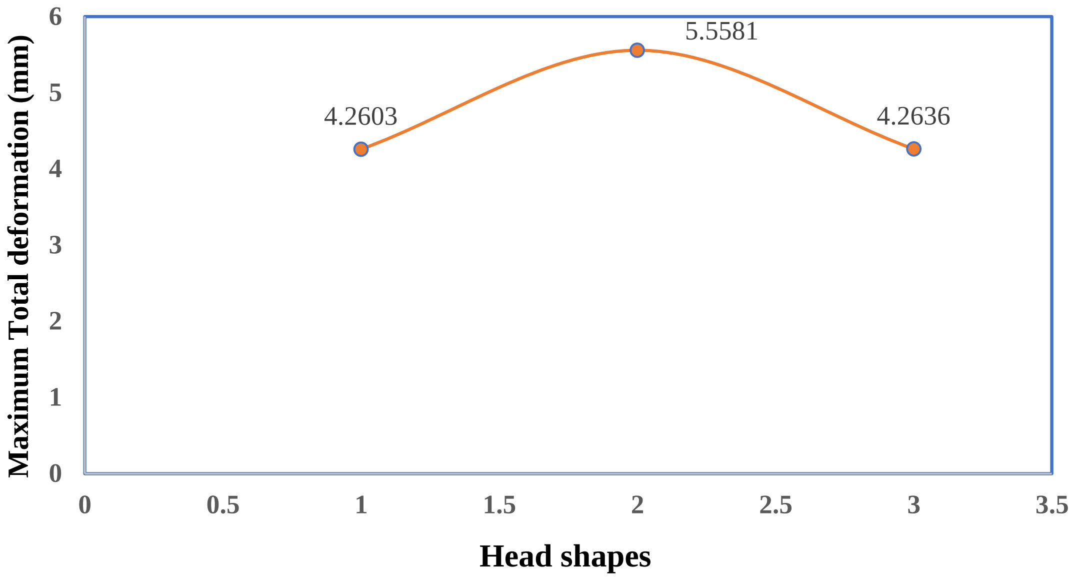


Fig. 6.8: Total deformation v/s different head shapes

It may be observed that from graphs (a) and (b) that torispherical head is much safer for the same material, internal pressure and internal temperature of 100°C because of the least value. So, in further analysis, torispherical head horizontal pressure vessel will be used.

5. Stress and deformation in different nozzle positions (a) one nozzle at head and one nozzle at 1/4th distance of the shell (b)both the nozzle at 1/4th of total length of the shell (c) both nozzles at the centre of the shell are shown through various graphs plotted below in fig.6.9 and fig.6.10.

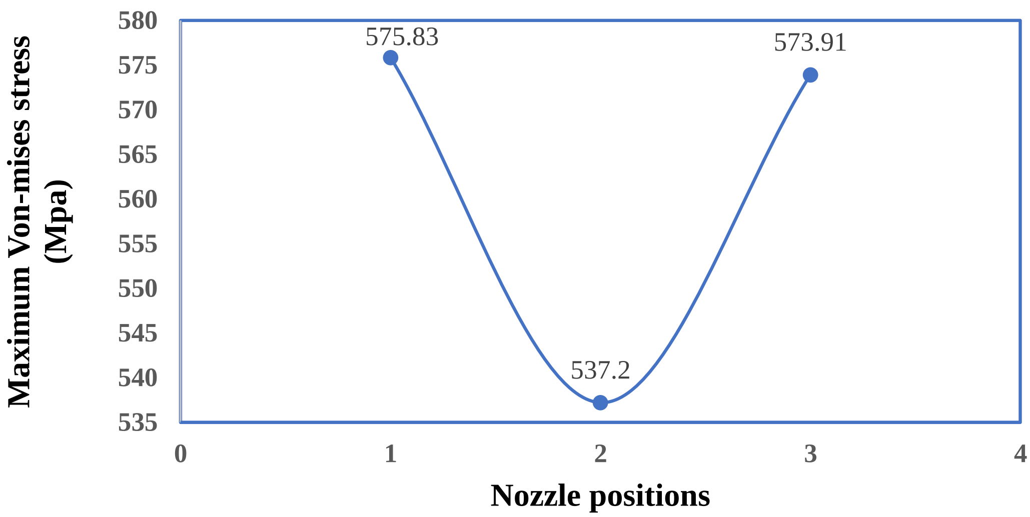


Fig. 6.9: Maximum von-mises stress v/s different nozzle positions

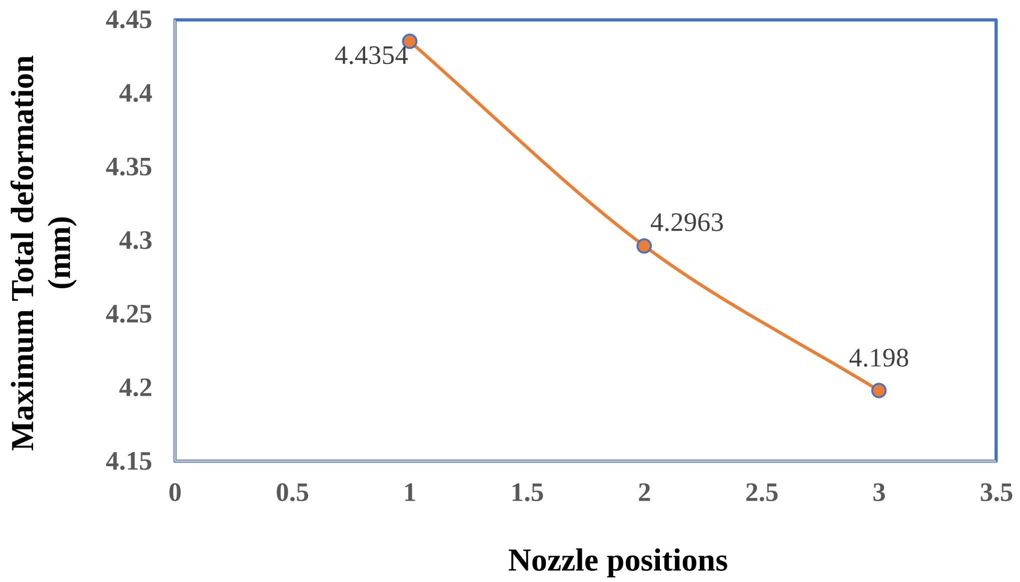


Fig. 6.10: Total deformation v/s different nozzle positions

Minimum of two nozzles are selected, one for inlet and other for outlet for different nozzle positions in horizontal pressure vessel. It may be observed from above graphs (c) and (d) that the maximum deformation induced when the nozzles are located at the centre of the shell is less. Hence, the nozzles are located at this position.

6. Stress and deformation in different support positions (a) at 1m (b) at 1.5 m and (c) at 2 m shown through various graphs plotted below in fig.6.11 and fig.6.12.

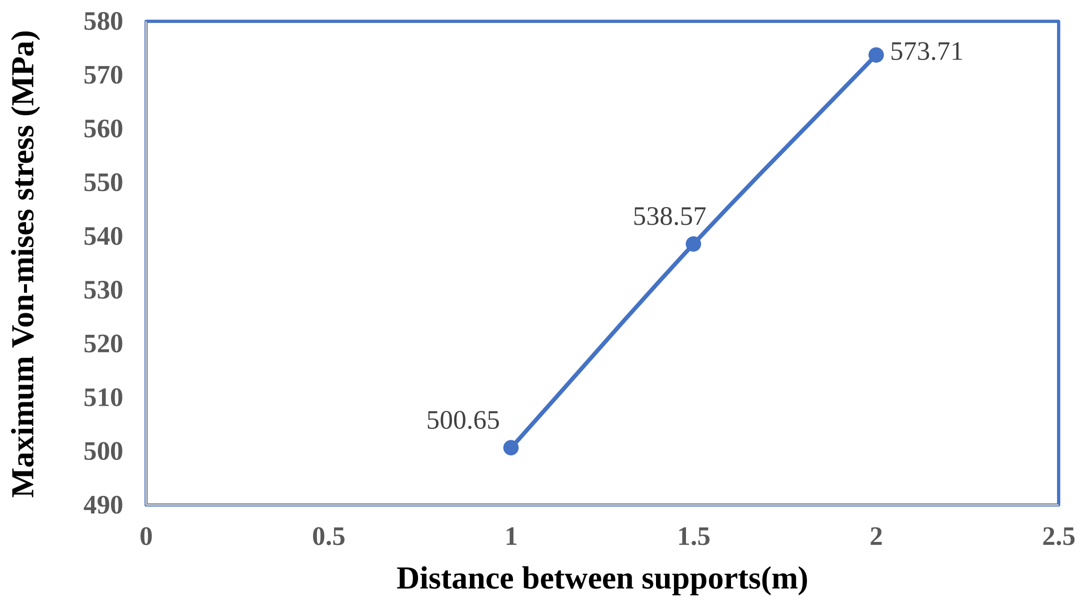


Fig. 6.11: Maximum von-mises stress v/s distance between supports

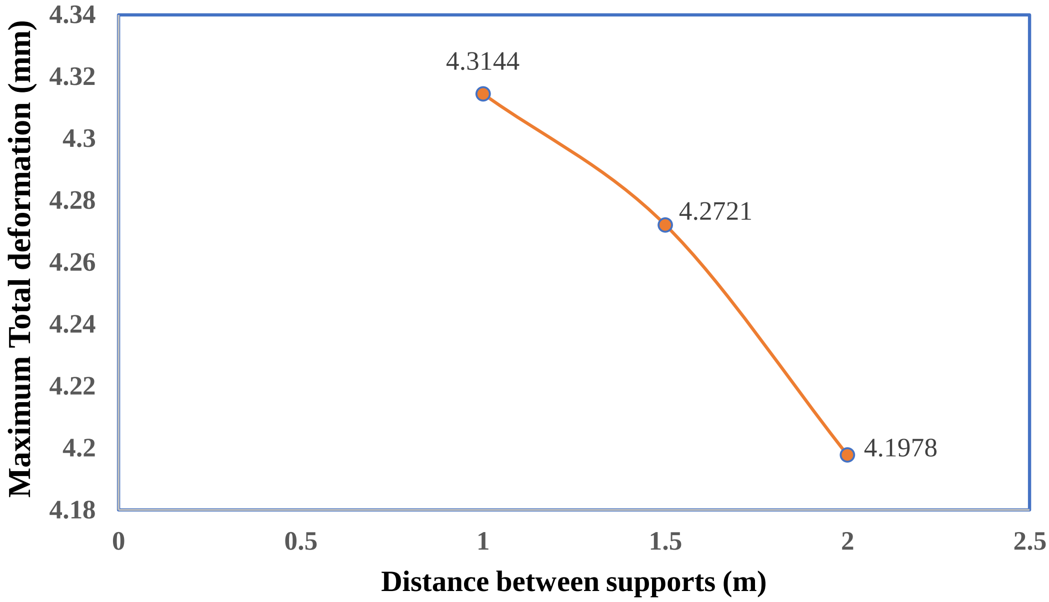


Fig. 6.12: Total deformation v/s distance between supports

Here the pressure vessels are available as beam for which two saddle supports are ideal. Distance between the saddles support is examined by three cases. It may be seen from graphs (e) and (f) that the least value of deformation is generated when the distance between the supports are at 2m. Hence, we observed that the least value of deformation will be generated when the supports are at extreme ends.

CHAPTER 7: CONCLUSION

7.1 Conclusions

This research work involves the design and optimization of a horizontal pressure vessel. Following are the analysis's ultimate conclusions:

1. Hemispherical, elliptical, and toruspherical are the three most common head forms. It can be shown that torispherical heads experience less stress overall than other head forms. For the same material and load, using a toruspherical head results in the least amount of stress.

2. In this investigation, three different nozzle placement combinations were used. They are: i) the inlet and outlet nozzles, which are situated at the top and side of the shell, respectively, at a distance of 25% of the length of the shell. ii) The inlet nozzle is located on the head and the outlet nozzle is situated at a distance of 25 percent of the total length of the shell from one end. iii) Inlet and outlet nozzles are located at the top and side of the shell, respectively, at distances of 50 percent of the total length of the shell from one end, or in the middle of the shell. Therefore, there is less tension when the inlet and outflow nozzles are placed in the middle of the shell.

3. Only two saddle supports are used in this investigation, and the impact of their separation is looked at. In this analysis, three instances are taken into account. i) A 1m gap between the supports' centres ii) 1.5 m distance centre of the supports and iii) 2 m distance between centre of the supports. It may be seen that the supports need to be sufficiently farther from the centre of the shell. That will result minimum value of stress and deformation.

7.2 Future scope of Work

- This pressure vessel structure was also modelled in ANSYS workbench with the purpose of performing the optimization using workbench's goal-driven optimization tool. But due to time constraint, this study could not be conducted.
- The analysis can be re-examined by modelling the thin cylindrical shell body of the pressure vessel using shell elements. Optimized results obtained from this shell model investigation can be compared and verified with the results obtained from the current work.
- Integrating ANSYS and CATIA. Optimization of design by pre-processing including meshing the finite element model, solving the model in ANSYS, followed by the optimization in ANSYS. **Altair Hyper mesh** provides superior meshing options with good element level control compared to all FEA software's available in the market. This will definitely give a much accurate solution since user can exploit the desired unique features limited to each software.
- Further analysis can be conducted on this pressure vessel by considering a few more design parameter such as radius of the cylindrical shell body, length of the cylindrical shell body, head thickness, head height, height of the nozzle openings etc.
- Metaheuristic based global optimization algorithms like genetic algorithm, Ant colony optimization algorithm, Differential Evolution and Simulated Annealing can be used to find the global optimum for the pressure vessel model used in this work.
- Shape or Topology Optimization can be implemented to remove the redundant material from specific components of the pressure vessel.

References

[1] Chattopadhyay, Somnath. *Pressure vessels: design and practice*. CRC press, 2004. https://doi.org/10.1201/9780203492468

[2] Hassan, Sulaiman, et al. "Design and optimisation of pressure vessel using metaheuristic approach." *Applied Mechanics and Materials*. Vol. 465, pp. 401-406. Trans Tech Publications Ltd, 2014.

[3] Sahu, Y. K., and S. Nagpal. "Design and optimization of a low pressure vessel. Res." *J. Eng* 6.7 (2017): 1-6.

[4] Saidpatil, Vishal V., and Arun S. Thakare. "Design & weight optimization of pressure vessel due to thickness using finite element analysis." *International Journal of Emerging Engineering Research and Technology* 2.3 (2014): 1-8.

[5] Widiharso, H. S., M. Tauviqirrahman, and Muchammad J. Jamari. "Thickness optimization of pressure vessel for minimum weight using finite element method (FEM)." *Int. J. Eng. Technol.* 8.6 (2016): 2676-2682.

[6] Bhinde, D. N., and S. Rajanarsimha. "FEA analysis of pressure vessel with different type of end connections." *J. Emerg. Technol. Innov. Res* 4.02 (2017): 95-99.

[7] Prasanna, P., J. Sushma, and K. P. Srikanth. "Stress analysis and design optimization of a pressure vessel using Ansys package." *Materials Today: Proceedings* 5.2 (2018): 4551-4562.

[8] Lathuef, Shaik Abdul, and C. K. 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* 1.2 (2012): 218-221.

[9] Khobragade, Rashmi, and Vinod Hiwase. "Design, and analysis of pressure vessel with hemispherical and flat circular end." *International Journal of Engineering Science and Computing* 7.5 (2017): 12458-12469.

[10] 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.

[11] Joship Kacmarcik, Nedelijko Vukojevic, "Comparison Of Design Method For Opening In Cylindrical Shells Under Internal Pressure Reinforced By Flush (Set-On) Nozzles". 2011

[12] 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

[13] 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

[14] 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

[15] 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

[16] Arman Ayobstress,"Analysis Of Torispherical Shell With Radial Nozzle", The Institution Of Engineers, Malaysia, Vol. 67, 2006

[17] V.N. Skopinsky, "Stress In Ellipsoidal Pressure Vessel Heads With Noncentral Nozzle," Nuclear Engineering And Design 198, Pp 317-323, 2000

[18] 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

[19] 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

[20] 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

[21] 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)

[22] 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

[23] 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.

[24] Drazan Kozak, Ivan Samardzic, “Stress Analysis Of Cylindrical Vessel With Changeable Head Geometry,” Science Bulletin, Serie C, Vol. XXIII

[25] Arunkumar, S., PR Eshwara Moorthy, and N. Karthik. "Design optimization of horizontal pressure vessel." *Materials Today: Proceedings* 26 (2020): 1526-1531.

[26] ASME B&PV code: Section VIII, Division I, Design and Fabrication of Pressure Vessels, ASME International, 2001.