

PAPER • OPEN ACCESS

Design and Analysis of Vertical Pressure Vessel using ASME Code and FEA Technique

To cite this article: S.J Niranjana *et al* 2018 *IOP Conf. Ser.: Mater. Sci. Eng.* **376** 012135

View the [article online](#) for updates and enhancements.

Related content

- [Design and Analysis of Boiler Pressure Vessels based on IBR codes](#)
B Balakrishnan and B Kanimozhi
- [Random lattice structures. Modelling, manufacture and FEA of their mechanical response](#)
G Maliaris, I T Sarafis, T Lazaridis et al.
- [Effect of Geometrical Imperfection on Buckling Failure of ITER VVPSS Tank](#)
Saroj Kumar Jha, Girish Kumar Gupta, Manish Kumar Pandey et al.

Design and Analysis of Vertical Pressure Vessel using ASME Code and FEA Technique

Niranjana.S.J¹, Smit Vishal Patel², Ankur Kumar Dubey³

¹Department of Mechanical and Automobile Engineering, Faculty of Engineering, Christ University, India 560074

²Department of Mechanical and Automobile Engineering, Faculty of Engineering, Christ University, India 560074

³Department of Mechanical and Automobile Engineering, Faculty of Engineering, Christ University, India 560074

niranjana.s@christuniversity.in, niranjana_mc2@yahoo.co.in

smit.patel@btech.christuniversity.in

ankur.dubey@btech.christuniversity.in

Abstract—In this project we are designing a pressure vessel using ASME section VIII and Division 2, designing a closed container to find the required thickness of the shell, head, nozzle and leg support. Uniform thickness assigned to the entire vessel, Modelling of the pressure vessel is carried out using Pro-e 2.0; meshing is carried out using Hypermesh 6.1. Here we used 2D Quad element for the meshing, Analysis is carried out using ANSYS Software 11 for two different cases, working pressure and Maximum operating pressure, fatigue analysis is carried out, and the result is 10^6 . Finally, theoretical validation is carried out for the entire model, And the results are within the limit.

Keywords— Pressure Vessel, Creo, HyperMesh6.1, ANSYS11, hyper view, Stress Concentration Factor.

1. Introduction

A pressure vessel is a device which has a pressure difference inside it by atmospheric pressure. As high operating pressures are a danger, utmost care should be taken while designing the pressure vessels. Any mechanical structure fails if there are stresses induced in them. The pressure vessel life under cyclic load is related to the number of cycles it is exposed to and to the intensity of the stress [9]. The pressure vessel is assumed to be a thin cylinder, and therefore the analysis follows the thin cylinder formulae. The modeling was done on a modeling software pro-e, and a finite element analysis was carried out to highlight the various points of stress concentration. As anticipated the highest stress value occurs at the junction of the nozzle attachment, to analyze the aspects of stress concentration which may develop when the end closure of a high-pressure vessel is attached to a conically shaped nozzle. The main reason for this occurrence is that the conical nozzle must be connected separately. This process would result in geometrical discontinuities between the pressure vessel and the nozzle at the point of attachment [5]. The solution for the value of stress at the connection of a cylindrical nozzle to an ellipsoidal shape pressure vessel. The stress calculations were carried out using finite element method, and a parametric model was developed [8]. Tests were carried out for various positions of the cylindrical nozzle on the pressure vessel along with multiple orientations of the connection instead of a central radial hole [2]. The tests carried out confirmed that the actual stress value of the maximum allowable internal pressure is different for various orientations of the attachment and states that further investigation is necessary to determine the most efficient connection.[6]



The accuracy of a finite element model depends on how the mesh is. If the mesh is coarse, the efficiency of the results decreases. At one point, we reach the point of diminishing returns where no matter how good the mesh is, it won't have a significant effect on the accuracy of the results. The mesh is said to be converged at this point. For all the models analyzed above, convergence has been observed as the mesh got refined [7]. From the result, the convergence seen is a monotonous one rather than an oscillating one. As the number of nodes and elements increases, the accuracy of the result also increases. From the analysis, the maximum stress occurs at the junction of the nozzle and pressure vessel. High-stress concentration is developed due to the abrupt change of the geometry and change in the stress. Symmetry is a significant factor than some nozzles as it is observed that peak stress for a symmetrical nozzle is very low and the stress increment factor also lowers. [1]

The experimental and analytical methods have been used to determine the stress intensity factor at sites of crack propagation on the internal and external surfaces of a cracked cylindrical pressure vessel. The results showed that stress intensity factor is proportional to the increase in the length of the crack. Strain gauges were used to measure the displacements and using empirical formulae stress intensity and correction factors were determined. The theoretical and experimental results were tallying. [3]

The finite element analysis based workbench is used for analyzing pressure vessel components. It discusses modeling methods for various parameters in a cracked pressure vessel. It also gives few rules for performing analysis using fem like 'starting with a simple design' and using, closed-form solutions for analysis. [4]

2. Design of Pressure Vessel

TABLE 1 - MATERIAL SPECIFICATION

Components	Material Grade
Flanges, Forgings	SA- 516 Gr 70
Shell, Dish End	SA - 516 Gr 70
Base Plate, Rib Plate	SA - 516 Gr 70

TABLE 2 – MATERIAL PROPERTIES

Tensile strength	484 Mpa
Yield strength	260 Mpa
Young's modulus	200 Gpa

A. Finding Thickness of Shell

The thickness of shell is detected using the below formulas, for circumferential stress and longitudinal stress the width of the shell must be higher to withstand the internal pressure and design pressure,[1]

$$t = \frac{PR}{SE - 0.6P} \quad \text{or} \quad P = \frac{SEt}{R + 0.6t}$$

$$t = \frac{PR}{2SE + 0.4P} \quad \text{or} \quad P = \frac{2SEt}{R - 0.4t}$$

**TABLE 3
OPERATING CONDITIONS AND DESIGN DATA**

Internal Design pressure=P	3.6 MPa
Internal Operating pressure=p	2 MPa
Design temp	250 ⁰ C
Operating temp	65 ⁰ C
Inside Diameter=Di	1300 mm
T/L – T/L	3000 mm

Type of Heads	Ellipsoidal(2:1)
Allowable stress=S	138MPa
Design No. of Cycles	1e5
Joint efficiency =E	1
Corrosion Allowance =C	1.5

Circumferential stress criterion

Checking for $0.385 \cdot S \cdot E$

$$S = 138$$

$$E = 1$$

$$0.385SE = 53.13 > 3.6$$

$$T_s = (P \cdot R) / (S \cdot E - 0.6 \cdot P) \text{ per UG-27 (c) (1)}$$

$$= (3.6 \cdot 650) / (138 \cdot 1.00 - 0.6 \cdot 3.6)$$

$$T_s = 17.226\text{mm} + C = 18.72\text{mm}$$

Nominal thickness ~20mm

The nominal thickness of the shell is assigned (UG-16), and for another loading (UG-22), it is the mating part for the head, nozzle, and flanges. [3]

A. Finding thickness of Ellipsoidal Heads

The nominal thickness of a semi-ellipsoidal form is found out using the following formula, to achieve uniform thickness throughout the vessel we choose 20mm thick.

$$t = \frac{PD}{2SE - 0.2P} \quad \text{or} \quad P = \frac{2SEt}{D + 0.2t}$$

$$T_h = (P \cdot D) / (2S \cdot E - 0.2 \cdot P) \text{ per UG-27 (c) (3)}$$

$$T_h = (3.6 \cdot 1300) / (2 \cdot 138 \cdot 1 - 0.2 \cdot 138)$$

$$T_h = 17.00087 + 1.5 = 18.5008\text{mm}$$

Nominal thickness ~ 20mm

B. Maximum Permissible/Allowable Working Pressure (MAWP)

Maximum permissible stress, is the stress that may be competently carried out on a vessel, it consists of a protection line along with the stress limit in a part when subjected to hundreds of stress cycles that could cause failure. The extreme permissible pressure in its typical operating position at a specific temperature at the top of the vessel is MAWP of the pressure vessel. Habitually when the design temperature is being calculated, the MAWP should be engraved on the nameplate. This pressure is established on calculations for every element of the vessel using nominal thicknesses exclusive of corrosion allowance. It is the foundation for creating the fixed constraints of pressure-regulating devices guarding the vessel.

MAWP, at Design Temperature,

Shell:

$$P_w = \frac{SdtEt}{Rc + 0.6t} \quad \text{or} \quad \frac{SdtEt}{Ro - 0.4t}$$

Head:

$$P_w = \frac{2SEt}{D + 0.2t} \quad \text{or} \quad \frac{2SEt}{D - 1.8t}$$

$$D_o = 1332\text{mm}$$

$$t_{sc} = 17.22\text{mm}$$

$$t_{hc} = 17.00\text{mm}$$

$$\text{MAWP for shell} = (138 \times 1 \times 17.22) / (1332 / 2 - 0.4 \times 17.22)$$

$$\text{MAWP for shell} = 3.61 \text{ Mpa}$$

$$\text{MAWP for head} = (2 \times 138 \times 17.0 \times 1) / (1332 - 1.8 \times 17.0)$$

$$\text{MAWP for head} = 3.605 \text{ Mpa}$$

3. FEA of Pressure Vessels

The 3D model and meshing are carried out in Pro-E 2.0, and HYPERMESH 6.1 software, 2D –Quad and Hexa8 elements are used for meshing, the total number of the element is 70,435. The analysis is carried out for the entire module using ANSYS software version 11, the following figure (Fig 1, 2) shows 3D modeling and Meshing of the vertical pressure vessel.

The analysis is carried out for operating pressure of 20bar and design pressure of 24 bar, by considering the uniform thickness of 20mm for the entire pressure vessel, shown in Fig 3, 4, 5.



Fig. -1 3D Model of Vertical Pressure Vessel.

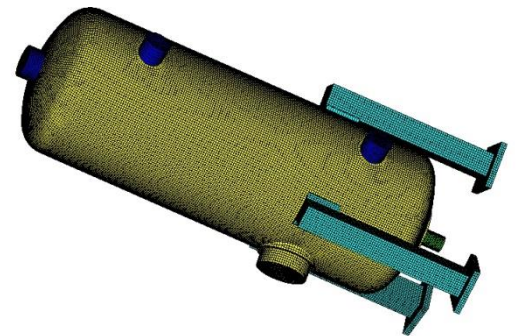


Fig.-2 Finite Element Model of Equipment

Model is analyzed for different element sizes, and size is selected wherein satisfactory accuracy is obtained having less computation time.

The 3D geometry model is meshed using 2D-Quad and Hexa 8 having an element size of 10mm. The total numbers of elements are 70,435 and Nodes 76,473.

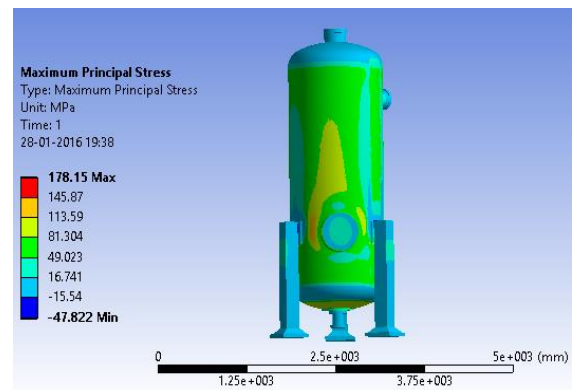
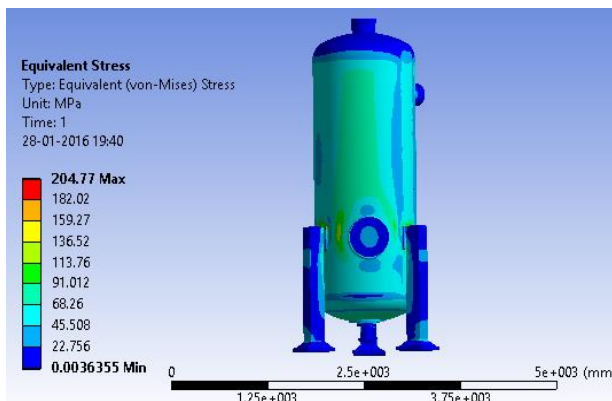


Fig.-3 Von-Mises Stress Plot details under an plot details

Fig.-4 Maximum principal Stress

Internal pressure of 20bar.

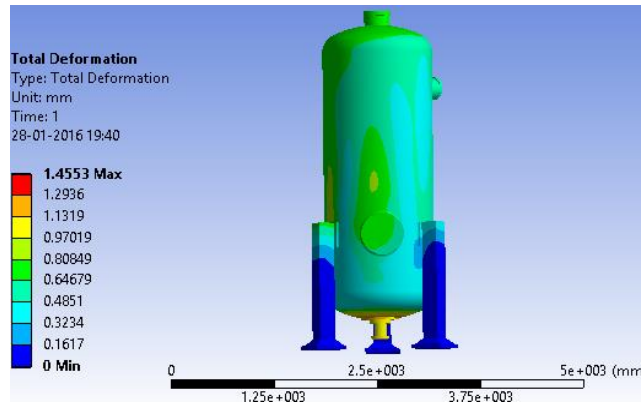


Fig.-5 Total Deformation Plot details

Maximum deformation of 1.4553 mm happens nearby the dish end and the cylindrical walls as shown in fig.-5.

4. Fatigue Life Calculation

By design data for a cyclic Pressure of 0MPa to 2MPa, the design number of cycles is 1×10^8 . According to point number 5.5.3.2 (ASME Section VIII, Division 2, Part 5, Point 5.5.3.2) the effective alternating stress amplitude for the kth cycle. [2]

$$S_{alt,k} = \frac{K_f \cdot K_{e,k} \cdot \Delta S_{p,k}}{2} \tag{5}$$

Where,

$\Delta S_{p,k} = 204.77$ Mpa.

K_f = Fatigue Strength reduction factor = 1.2 (ASME Section VIII, Div. 2, Table 5.11) [2]

$K_{e,k}$ = Fatigue Penalty Factor = 1

(Since $\Delta S_{n,k} < \Delta S_{PS}$; i.e. $204.77 < 500$ MPa (max [3S, 2Sy]))

After solving this we get,

$S_{alt,k} = 122.862$ Mpa.

To find the number of the design cycles following formula is applied. The number of design cycles ‘N’ can be computed from the below Equation or Table 3, based on the stress amplitude, Sa which is determined by Part 5 of this Division [11-12].

$$N = 10^X \quad \text{Where, } X = \frac{C_1 + C_3Y + C_5Y^2 + C_7Y^3 + C_9Y^4 + C_{11}Y^5}{1 + C_2Y + C_4Y^2 + C_6Y^3 + C_8Y^4 + C_{10}Y^5} \tag{6}$$

$$Y = (S_a / C_{us}) + (EFC / ET) \tag{7}$$

Sa = Stress Amplitude.

N = Number of cycles.

ET= Material modulus of elasticity at the cycle temperature.

EFC= Modulus of elasticity used to establish the design fatigue curve.

CUS = 6.894757 for units of stress in MPa. (Conversion factor)

For the vessel materials of construction, the coefficients C_i and the modulus EFC are taken from Smooth bar fatigue curve for carbon steel $\sigma_{uts} \leq 552$ MPA are listed in table 3.F.1.[11],[2]

TABLE 4
COEFFICIENT OF FATIGUE CURVES (ASME SECTION VIII, DIVISION 2, PART 3)

Coefficients Ci	48 ≤Sa<214 (Mpa)
1	+2.254510E+00
2	-4.642236E-01
3	-8.312745E-01
4	8.634660E-02
5	2.020834E-01
6	-6.940535E-03
7	-2.079726E-02
8	+2.010235E-04
9	+7.137717E-04
10	+6.2300053E-5
11	+5.832781E-04

After solving the above equations we get,

$$N = 10^{5.887}$$

$$\text{Fatigue damage factor} = 10e5 / 10e5.88 = 0.20044$$

This factor is much less than unity.

The design is safe.

Calculation

Compute Primary Membrane Stress [S]:

$$S = P / (E \times \ln((2 \times t + D_i) / (D_i))) \tag{9}$$

$$S = 2 / (1 \times \ln((2 \times 20 + 1300) / (1300)))$$

$$S = 65.99 \text{ MPa}$$

Sample calculation for the Intensified Stress Amplitude [Sa]:

$$[Sa] = S \times 3.3 / 2 \tag{10}$$

$$[Sa] = 65.99 \times 3.3 / 2$$

$$[Sa] = 108.8 \text{ Mpa}$$

Stress Factor used to compute [Y]:

$$[Y] = (Sa / C_{us}) \times (EFC / Et) \tag{7}$$

$$[Y] = (108.8 / 6.89) \times (195e3 / 200e3)$$

$$[Y] = 15.69$$

Compute from Equation [X]:

$$[X] = (C_1 + C_3 \times Y + C_5 \times Y^2 + C_7 \times Y^3 + C_9 \times Y^4 + C_{11} \times Y^5) / (1 + C_2 \times Y + C_4 \times Y^2 + C_6 \times Y^3 + C_8 \times Y^4 + C_{10} \times Y^5) \tag{6}$$

$$[X] = 5.887$$

TABLE 5
ALLOWABLE NUMBER OF CYCLES, N_k

ET (MPa)	EFC (MPa)	Salt, k (MPa)	X	N _k (cycle)	nk (designed cycles)
200E3	193E3	122.86	5.887	7.70E5	1e5

Calculation of fatigue damage factor

The actual number of recurrences of the cycle is set to the cyclic life requirements delivered in the User's Design description, 1e4 cycles. The fatigue damage is then calculated by

$$Df, k = nk / Nk \leq 1 \tag{8}$$

The results are calculated for fatigue damage for a limiting region of 0.2511. The fatigue damage is well below the allowable level of 1.0, satisfying the requirement. It is mandatory to pass NSF certification for the pressure vessels for this it must provide a minimum service life of 100,000 cycles. However, some manufacturers surpass this standard and offer tanks that offer performance as long as 250,000 cycles.

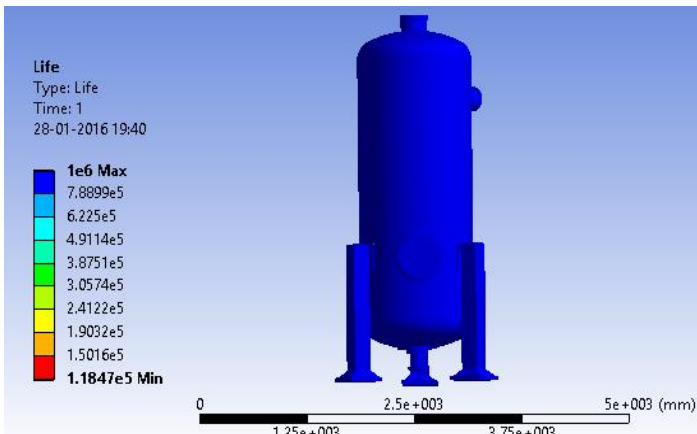


Fig.-6 Shows the fatigue life details.

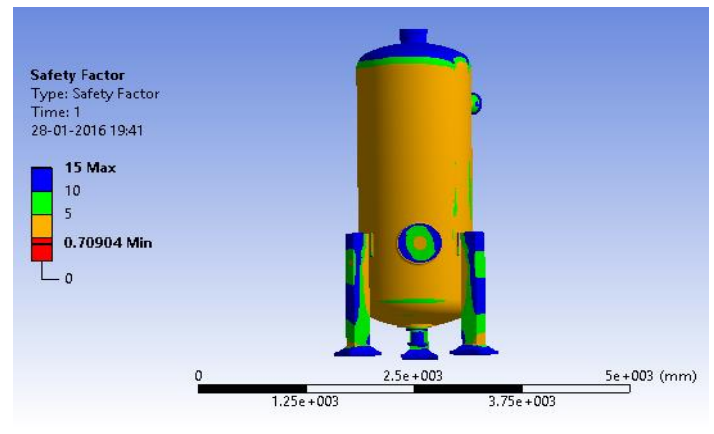


Fig.-7 Safety factor details.

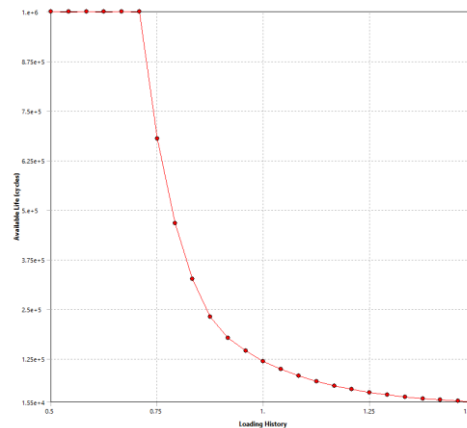


Fig.-8 S-N curve details

5. Evaluation of Longitudinal and Hoop Stresses of Pressure Vessels

The stresses acting on pressure vessel are principal stresses, hoop stress or circumferential stress (σ_H) and longitudinal stress, (σ_L) as shown in (Fig. 9). Circumferential stress is the stress which is set up in resisting the bursting effect of the applied pressure and can be most conveniently treated by considering the equilibrium of the cylinder.

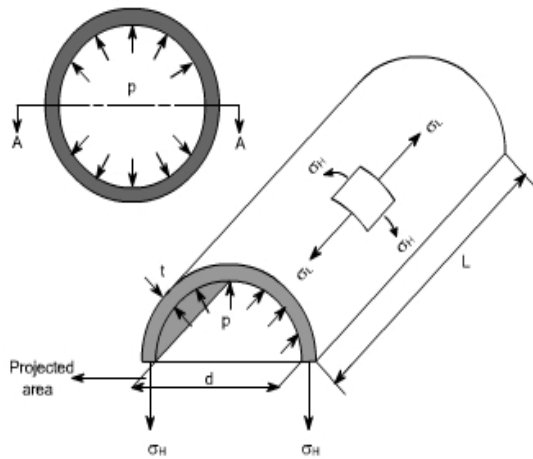


Fig.-9 Hoop or circumferential stress acting on the cylinder [11] details

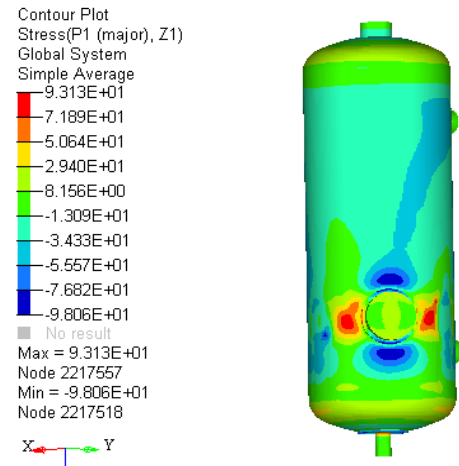


Fig.-10 Hoop Stress plot

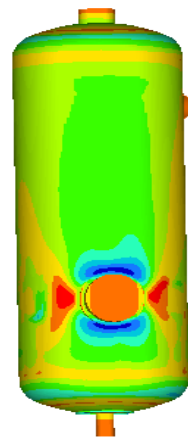
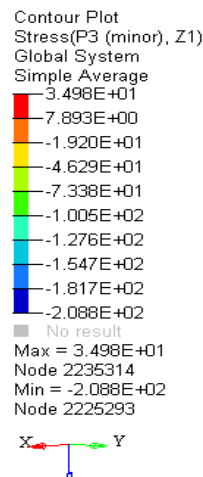


Fig.-11 Longitudinal Stress plot details.

Hoop stress or circumferential stress can be calculated using the formula: The formulae in ASME Section VIII, Division 1, paragraph UG-27,

$$\text{Hoop Stress} = \sigma_H = p \cdot r_i / (t \cdot E \cdot 0.6 \cdot p) \tag{10}$$

Where,

p = internal working pressure.

r_i = internal radius.

t = wall thickness of the pressure vessel.

E = Joint efficiency.

$$\sigma_H = (2 * 1300/2) / (20 * 1 - 0.6 * 2)$$

$$\sigma_H = 69.149 \text{ Mpa.}$$

Longitudinal Stress can be calculated using the formula:

$$\text{Longitudinal Stress} = \sigma_L = p * r / (2t E - 0.6 * p) \quad (11)$$

$$\sigma_L = (2 * 1300/2) / (2 * 20 * 1 - 0.6 * 2)$$

$$\sigma_L = 33.1632 \text{ Mpa.}$$

6. Conclusion

In this project, a pressure vessel is designed by the stipulation recommended by ASME. Three-dimensional modeling is carried out in Pro-e. The analysis is carried out on the pressure vessel to verify the strength by observing the result it's clear that for the initial iteration for 18mm thickness the stress values were higher 270.6MPa than the yield stress 260MPa of the material. Thereby call for optimization was essential, and the thickness of the pressure vessel was increased to 20mm. And from the analysis report, Von-misses stress was found to be 204.7 MPa, which is less than yield stress and the factor of safety value is 1.25 obtained. Stresses on the leg support are found to be 133MPa, and its well below the yield stress, the highest value for stress was developed around the nozzle where the interaction between shell and nozzle attachment of the vessel.

Fatigue analysis is carried out for the entire structure of the pressure vessel, and from the fatigue analysis report, it is clear that the fatigue life cycle is more than the required period. Thereby we conclude that examined parts for fatigue are well within the permissible limit set by ASME design code. The fatigue damage code factor experienced is less than unity, so the design is safe, by this safety of pressure vessel is established well before production, and preventive measures are taken to ensure a fatal accident doesn't take place.

REFERENCES

- [1] Pravin Narale¹, Prof. P .S.Kachare², “*Structural Analysis of Nozzle Attachment on Pressure Vessel Design*” International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 2, Issue4, July-August 2012, pp.1353-1358.
- [2] Shyam R Gupta¹, Ashish Desai², “*Optimize Nozzle Location for Minimization of Stress In Pressure Vessel,*” Innovative Research in Science & Technology| Vol. 1, Issue 1, June 2014| ISSN(online): 2349-6010.
- [3] Mundhe N.D.¹, Utpat Abhaya.², “*Analysis of Cracked Cylindrical Pressure Vessel by using Experimental approach,*” International Journal of Latest Trends in Engineering and Technology (IJLTET), Vol. 2 Issue 3 May 2013.
- [4] Michael A. Porter¹, Dennis H. Martens², Pedro Marcal³, *On Using Finite Element Analysis for Pressure Vessel Design.*
- [5] Pravin B.Sonawane¹, P.G.Damle², “*Design And Analysis of Top Conical End Closure Nozzle Junction And Pressure Vessel,*” International Journal of Engineering Sciences & Research Technology, Value: 3.00, August 2016.
- [6] V.N. Skopinsky¹, A.B. Smetankin², “*Modeling And Stress Analysis Of Nozzle Connections In Ellipsoidal Heads Of Pressure Vessels Under External Loading,*” Int. J. of Applied Mechanics and Engineering, 2006, vol.11, No.4, pp.965-979.
- [7] Zainul Huda¹, Muhammad Hani Ajani², “*Evaluation of Longitudinal and Hoop Stresses and a Critical Study of Factor of Safety (FoS) in Design of a Glass-Fiber Pressure Vessel*”, World Academy of Science, Engineering and Technology, International Journal of Civil, Environmental, Structural, Construction and Architectural Engineering Vol:9, No:1, 2015.
- [8] B.S.Thakkar¹, S.A.Thakkar², “*Design of Pressure Vessel Using ASME Code, Section VIII, Division I*”, International Journal of Advanced Engineering Research and Studies, IJAERS/Vol. I/ Issue II/January-March, 2012/228-234.

- [9] Kirtikumar Tamboli¹, “*Fatigue Analysis of Pressure Vessel by FEA Techniques*,” International Journal of Engineering Trends and Technology (IJETT) – Volume 13 Number 1 – Jul 2014.
- [10] Joseph Mutava¹, Onesmus Muvengi², Kenneth Njoroge³, John Kihiu⁴, “*Solutions to Pressure Vessel Failures: A Review*,” International Journal of Science, Engineering and Technology Research (IJSETR) Volume 6, Issue 2, February 2017, ISSN: 2278 -7798.
- [11] Open source.
- [12] Kiran K¹, A M Mulla², Manoj³ and Umesh C J⁴, “Investigation of baffle spacing effect on shell side heat transfer characteristics in shell and tube heat exchanger using computational fluid dynamics,” Elixir International Journal, Volume -3, 2014.
- [13] Apurva R. Pendbhaje¹, Mahesh Gaikwade², NitinDeshmukh&RajkumarPatil³“*Design & Analysis of Pressure Vessel*,” International Journal of Innovative Research in Technology & Science, Vol.2 Number 3.
- [14] A. Devaraju¹, k. Pazhanivel²” *A STUDY ON STRESS ANALYSIS FOR DESIGN OF PRESSURE*” International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-11, Nov.-2015.
- [15] Prof. Mr. Amol Mali¹, Mr. Hemant Bhosale², Mr. Dilpreet Singh Bedi³, Mr. Akash Modasara⁴” *Design of Vertical Pressure Vessel using PV Elite software*,” International Research Journal of Engineering and Technology, Volume: 04 Issue: 05 | May -2017
- [16] Mane S.S ¹, Prof. Wankhede P.A²” *THE DESIGN OF VERTICAL PRESSURE VESSELS SUBJECTED TO APPLIED FORCES AND VIBRATIONAL CONDITIONS*,” International Journal of Mechanical Engineering and Technology, ISSN 0976 – 6340(Print), ISSN 0976 – 6359(Online) Volume 3, Issue 2, May-August (2012).
- [17] Prof Mr.Aniket Kolekar¹, Mr.Akshay Natak², Mr.Kiran Navratne³, Prof.Mr.Amol Mali⁴ “recent advancement in shape memory alloys ” international research Journal of Engineering and Technology (IRJET) Vol-04 Issue-04 April -2017 Page No. 2120-2125.
- [18] Hossein.kazem¹, Mahmood.minavand², “Seismic evaluation and strengthening of vertical and horizontal pressure vessels in a refining complex,” International Journal of Advanced Structural Engineering, Vol. 3, pp. 101-110, 2011.
- [19] Chinna Rosaiah Unnava¹, Ch. Siva Ramakrishna², “Design and analysis of spherical shell with Radial nozzle in pressure vessels” International Journal of Advanced Engineering Research and Studies, pp.52-54, 2013.