iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 10 **Issue:** VII **Month of publication:** July 2022

DOI: https://doi.org/10.22214/ijraset.2022.46076

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue VII July 2022- Available at www.ijraset.com

Design & Finite Element Analysis of Pressure Vessel

Kunal Khadke[1], Dr.Dinesh Chawde[2]
[1]2nd Year M.Tech, CAD/CAM & Robotics, [2]Professor, Department of Mechanical Engineering, KJ Somaiya College of Engineering, Mumbai, India

Abstract: This document - Pressure vessels are of immense importance in most of the industries today & are drastically used in many fields such as chemical, petroleum, military industries as well as in nuclear power plants. Catastrophic accidents can occur due to rupture of pressure & as a result they should be designed & analysed with immense care & precision. The exact estimation of stresses due to the applied mechanical & thermal loads are the common problems faced by any engineer while designing the vessel. This paper aims to design of pressure vessel using ASME Code Book, accordingly model the vessel in Solidworks & carrying out the finite element analysis for the Pressure vessel using ANSYS. The critical parameters of pressure vessel that are taken into account includes internal pressure, seismic, wind loads & operational loadings. The paper focuses upon the design of thin walled pressure vessel particularly a tall vessel where the vessel is designed using manual calculations & performed FEA to determine the stress in the vessel due to multiple loadings. It consists of design of thin walled pressure vessel made up of homogeneous material where the stresses are plotted along with the contour plots & have verified the FEA results with the analytical solution.

Keywords: **pressure vessel, design, ASME, finite element analysis, ANSYS**

I. INTRODUCTION

The pressure vessel is a vessel designed to contain a gas or liquid at a pressure significantly different from its ambient pressure. Ultra high pressure units are expected to be widely applied in nuclear industry, power generation sector, offshore facilities & chemical industries. Stress analysis & evaluation of stress in pressure vessels is important for solving basic design problems. Each pressure vessel must operate at the design temperature and pressure, which are the safety limits of the pressure vessel. This is because accidental leakage or leakage of contents may pose a hazard to the environment. Some of the well-known standards are ASME BPVC (American Society of Mechanical Engineers Boiler and Pressure Vessel Code), Title VIII, and API (American Petroleum Industry) Pressure Vessel Inspection Code 510. The paper focuses upon the design of thin walled pressure vessel particularly a tall vessel where the vessel is designed using manual calculations & performed FEA to determine the stress in the vessel due to multiple loadings. The main aim of this report is to investigate and determine the stress analysis through the wall of cylindrical pressure vessels – thin wall pressure vessels subjected to different loads

II. PROBLEM DEFINITION

Improper design and construction, and irregular testing and inspection endanger the safety of pressure vessels. When fluids are stored under pressure, it is more likely to rupture and leak. Defects such as corrosion, loss of thickness, mechanical and metallurgical defects, cracks, mechanical deformation etc. will adversely affect the operation of this device. Pressure vessels that do not comply with standard codes can be very dangerous. In fact, there have been many fatal accidents in the entire history of their operation and development. As a result, the goal of the paper is to analyse the stress concentration generated in the pressure vessel and predict the maximum load that the vessel can take considering all of the design parameters & environmental factors thereby avoid catastrophic failures in the future

III. OBJECTIVES

The objectives of the proposed model are summarized below are to perform Stress analysis of thin wall pressure vessel using ANSYS which will include modelling & FEA analysis the pressure vessel along with the appropriate supports in Solidworks & ANSYS respectively using design calculation for a particular design pressure Secondly it also aims to analyse the contour plot the Maximum Principal Stress, maximum tensile stress & Von Mises stress in the pressure vessel upon loading.

It also gives an idea to understand the critical areas of failure & accordingly provide reinforcement to the areas. When a pressure vessel has complex geometry or loading conditions such that traditional methods (i.e. hand calculations) are inadequate in accurately evaluating the vessel, finite element analysis can be used to ensure the design is acceptable.

IV. LITERATURE REVIEW

Senthil Anbazhagan in this paper shows an analytical approach based wind and seismic design recommendations for vertical tall process column & focuses on how the wind and seismic analysis is a fundamental requirement for equipment design in the oil and gas industry. In this article, column heights are designed to withstand external seismic and wind action and some important design steps to consider to avoid column breakage. M. A. Khattak depicts in his paper the common root causes of pressure vessel failures & conclude a brief review of various pressure vessel failure mechanisms is presented, taking into account the toughness of heat-aged samples, the effects of heat aging on the adsorbed hydrogen and plastic regions of the crack tip of welded steel, and the fatigue cracking mechanisms of welds. Effect of fatigue crack growth in austenitic steel and thermally aged samples in welded steel area also observed. G.S. Vivek paved the way for design and analysis of vertical pressure vessel where a case study was conducted by performing a linear static analysis on the vertical pressure vessel for stress analysis performed in accordance with the ASME code, and through this analysis, the FEA analysis result showed that the equivalent stress concentration was under various pressure conditions under the maximum allowable stress SA 516 Gr 70 in pressure. Apsara C. Gedam depicts stress analysis of pressure vessel with different end connections on the journal published by her where the goal of paper is to compare stress distribution in the pressure vessel for different end connections viz. hemispherical, flat circular, standard ellipsoidal and dished shaped pressure vessel heads. Norliza Rahman enlightens an interactive short cut method for pressure vessel design based on ASME where he proposed interactive decompression vessel design method based on ASME standards. In this article, it has proposed an interactive, abbreviated method for designing pressure vessels to determine the best and most reliable application of the system. The system has been proven to pass testing and validation using two different approaches. Testing and learning from last year's project shows a comparison between manual ASME code calculations and iPVD calculations

V. METHODOLOGY

When a pressure vessel has a complex shape or loading conditions that make it unsuitable for accurate evaluation of the vessel by conventional methods & manual calculation, finite element analysis can be used to determine if the design is acceptable. There are several types of load conditions that can be analysed using FEA. These include internal pressure, external pressure, self-weight, thermal loads, cyclic loads, shocks and shocks, seismic loads, wind loads, vibration loads, and loads from external nozzles. This report enlightens about how boundary conditions can be implemented for particularly a tall vessel which is subjected to structural loadings & how the stress is generated in different parts of the vessel

VI. PROPOSED WORK

A tall vessel has been designed using Solid works where the tower is divided into six sections. An ellipsoidal head at the top followed by cylindrical shells & flanges mounted on each shells. At the lower section there is a conical shell & lastly there is one more ellipsoidal head to bottom of the vessel & all of these shells are supported by skirt support.

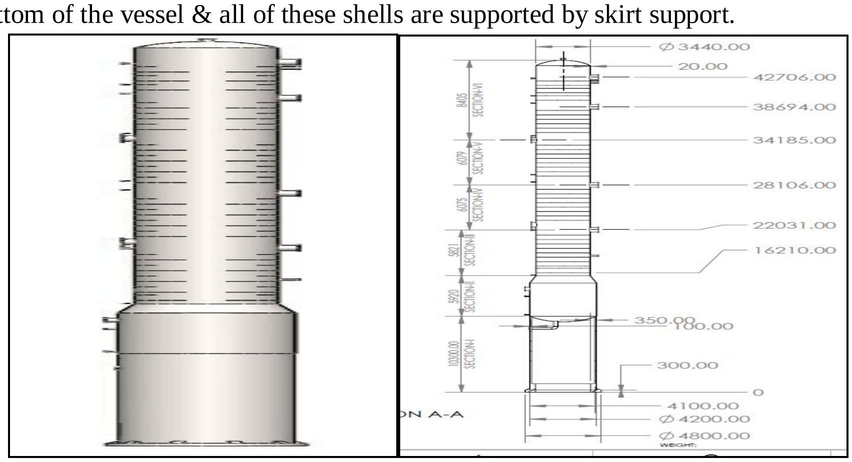


Figure 1. Geometry of tall vessel

Table I. Parameters of the design

ASME Sec. Viii, Div. 1 Latest		
P	Design Pressure	1.5 MPa
T	Design Temperature	250^0 C
E	Weld Joint Efficiency	0.85
Ca	Corrosion Allowance	1.5 mm
Material Of Construction		
Shell	Sa 515 Gr. 70 (Tensile strength) = 400MPa	
Skirt	Sa 515 Gr. 70 (Tensile strength) = 400MPa	
Construction Details		
D	Shell Id	3.4 m
H	Overall Column Height	42.6 m
H	Skirt Height	10.2 m
Type of dished end : 2 : 1, Ellipsoidal Insulation: Yes (HOT) 100 mm. Internals :40 Trays (Valve Type)		

A. Elliptical Head Calculation

Internal Pressure

Required Thickness due to Internal Pressure [tr]:

= (P*Do*Kcor)/(2*S*E+2*P*(Kcor-0.1)) per ASME Code Section VIII Division 1

= (1.5*4.0*0.998)/(2*137.9*1.0+2*1.5*(0.998-0.1))

= 21.4985 + 3.1750 = 24.6735 mm

External Pressure

Calculation for Maximum Allowable External Pressure (MAEP):

Tca	OD	D/t	Factor A	B
21.825	4.00	183.28	0.0007578	75.76

EMAP = B/(K0*D/t) = 75.764/(0.9 *183.2765) = 0.4593 N./sq.mm.

P = 1.67 * External Design pressure for this head.

Max. Allowable Working Pressure at given Thickness, corroded [MAWP]:

= ((2*S*E*t)/(Kcor*D+0.2*t))/1.67 per Appendix 1-4 (c)

= ((2*137.9*1.0*21.8249)/(0.998*3.9563+0.2*21.8249))/1.67

= 0.912 N./sq.mm.

Maximum Allowable External Pressure [MAEP]

= 0.912 N./sq.mm

B. Cylindrical Shell

Required Thickness due to Internal Pressure [tr]:

= (P*Do*Kcor)/(2*S*E+2*P*(Kcor-0.1)) per

= (1.5*3.4*0.997)/(2*137.9*1.0+2*1.5*(0.997-0.1))
= 18.2669 + 3.1750 = 21.4419 mm.
Results for Maximum Allowable External Pressure (MAEP):

Tca	OD	SLEN	D/t	L/D	Factor A	B
21.825	4.00	4.73	183.28	1.1823	0.00045	45.72

EMAP = (4*B)/(3*(D/t)) = (4*45.718)/(3*183.2765) = 0.3326 N./sq.mm
Maximum Allowable External Pressure [MAEP] = 0.3326 N. /sq.mm

C. Wind Load Analysis

Factor of Importance	1.000
Static & Dynamic Load factor (Gh, Gbar)	1.218
Shape factor for the vessel	0.632
Average speed of wind	31.3 m/sec
Category of exposure	C
Alpha value from table	7.0000
Zg value from table	900.0000
Do value from table	0.0050

D. Calculation for the First Element

Factor of roughness = 1.0
Values [cf1] and [cf2] as H / D < 25.0
CF = CF1 + (CF2-CF1) * (H/D - 7.0)/(25.0 - 7.0) = 0.6 + (0.7- 0.6) * (12.828 - 7.0)/(25.0 - 7.0) = 0.632
Value of Alpha, Zg is taken from reference tale [Alpha, Zg]
For category of exposure C:
Alpha = 7.0, Zg = 274.32 m.
Height of considered section [z]
= Centroid Height + Base Height = = 4.75 + 0.0 = 4.75
z = Max (4.572, 4.75) = 4.75 m.
Coefficient of pressure velocity [kZ]:
= 2.58(z/zg)^(2/Alpha) : z is Elevation of First Element = 2.58(4.75/900) ^ (2/7.0)
= 0.81
Ratio of height to diameter:
= Maximum Height (length) ^2 / Sum of Area of the Elements = 149.285(^2)/1737.224 = 12.828
Frequency of Vibration = 1.196 Hz
Outer Diameter of the vessel =3 m.
Damping factor of Vibration (Operating) Beta = 0.01
For category of terrace C
S = 1.0, Gamma = 0.23, Drag Coeff. = 0.005, Alpha = 7.0
Compute [fbar]
= 10.5 * Frequency (Hz) * Vessel Height (ft) / (S * Vr(mph)) = 10.5 * 1.196(Hz) * 149.285(ft)/S * 1.0(mph)
= 26.777
Calculation for wind pressure [qz]
Factor of importance: I = 1.0
Wind Speed = 31.292 m/sec Converts to 70.0 mph
qz = 0.00256 * kZ * (I * Vr)^2 = 0.00256 * 0.81*(1.0 * 70.0)^2 = 10.157 psf
Converts to: 486.292 N./m^2
Force on the First Element [Fz]
= qz * Gh * CF * Wind Area = 486.292 * 1.218 * 0.632 * 45.589 = 17085.02 N.
Using the above calculations for any element, load for wind loads at various heights are calculated & stated in the figure 2 below

Wind Height m.	Wind Diameter m.	Wind Area m²	Wind Pressure N./m²	Element Wind Load N.
4.75	4.8	45.5887	486.292	17085
9.7	4.8	1.91952	596.335	882.156
11.9	4.8	19.1952	632.201	9352.12
14.8337	4.43924	8.52123	673.284	4421.43
17.82	4.08	16.316	709.509	8921.38
20.7305	4.08	7.42784	740.85	4240.86
23.141	4.08	12.237	764.503	7209.66
26.141	4.08	12.237	791.599	7465.18
29.141	4.08	12.237	816.555	7700.54
32.141	4.08	12.237	839.739	7919.17
35.141	4.08	12.237	861.424	8123.67
38.141	4.08	12.237	881.824	8316.06
41.641	4.08	16.316	904.224	11369.7
44.4901	4.08	6.83732	921.485	4855.52

Figure 2. Wind loads at different sections of tower

E. Earthquake Analysis Results

The UBC Zone Factor for the Vessel is 0.3000
The Importance Factor as Specified by the User is 1.000
The UBC Frequency and Soil Factor (C) is 1.408
The UBC Force Factor as Specified 3.000
The UBC Total Weight (W) for the Vessel is 1029194.5 N
The UBC Total Shear (V) for the Vessel is 144936.9 N.
The UBC Top Shear (Ft) for the Vessel is 8484.4 N

Distance to Support m.	Cumulative Wind Shear N.	Earthquake Shear N.	Wind Bending N.mm.	Earthquake Bending N.mm.
4.75	107862	144937	2.648E+09	4.546E+09
9.7	90777.5	142099	1703791616	3.182E+09
11.9	89895.3	139710	1667642496	3.126E+09
14.86	80543.2	133587	1326627328	2.579E+09
17.82	76121.8	130016	1176168064	2.326E+09
20.7305	67200.4	120380	889407744	1824758272
23.141	62959.5	115277	770849088	1610105856
26.141	55749.9	105891	592712704	1278219776
29.141	48284.7	95289	436597632	976327168
32.141	40584.1	83470.2	303240320	708079552
35.141	32665	70434.6	193322096	477128640
38.141	24541.3	56182.4	107477864	287126016
41.641	16225.2	40713.4	46303224	141723408
44.141	4855.52	18195.3	4124633	23858292

Figure 3. Bending moments due to seismic & wind loads

F. FEA: Boundary conditions

The boundary conditions are applied in case of the maximum load conditions where the vessel is subjected to high seismic & wind load. Along with these external factors, the vessel is also subjected to load due to its own self weight & the hydrostatic pressure of the liquid present inside the vessel situated at height from section VI to section II. Internal pressure of 1.5MPa is applied on to the inner walls of the vessel. Accordingly, the bending moments due to wind & seismic loads are applied at the particular heights as shown in table

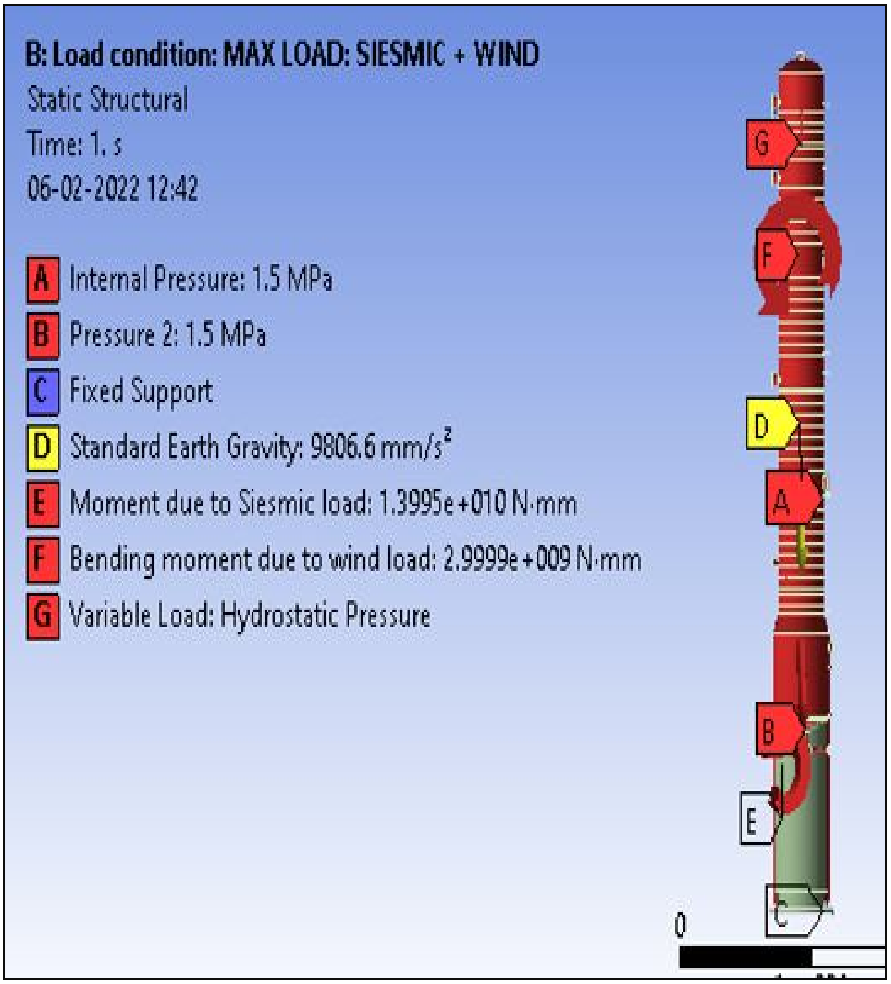


Figure 4. Defined boundary conditions

Parameter	Magnitude	Location
Internal Pressure (P_i)	1.5 MPa	Inner walls of the cylinder Normal direction to the surface
Fixed support	-	Base Plate
Standard Earth Gravity – Dead Weight of the vessel	309390 kg	At the centre of gravity
Variable Load: Hydrostatic Pressure	-Filled with water	Fluid is filled from section VI to section II
Bending Moment due to wind Load	30,00,000 N-mm	At the Centre of gravity of the vessel
Bending Moment due to Seismic Load	1,34,00,000 N-mm	At $\frac{2*H}{3}$

Table II. Location & Magnitude of boundary conditions

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue VII July 2022- Available at www.ijraset.com

G. FEA Results

After pre-processing, meshing & setup of the model in ANSYS, the setup is solved w=using ANSYS Structural Workbench solver. After observing the contour plot for the factor of safety the minimum factor of safety (0.9) is observed at the junction of conical shell & cylindrical shell due to bending loads because of the seismic & wind loads. Fig. 5 clearly shows the factor of safety at the nozzle openings where there is removal of material in the shells for welding the nozzle. A significant factor of safety (more than 1.5) is observed at each of the nozzle openings. So it can be concluded that the thickness of nozzle at the nozzle opening is adequate enough to sustain the loads ate maximum loading condition

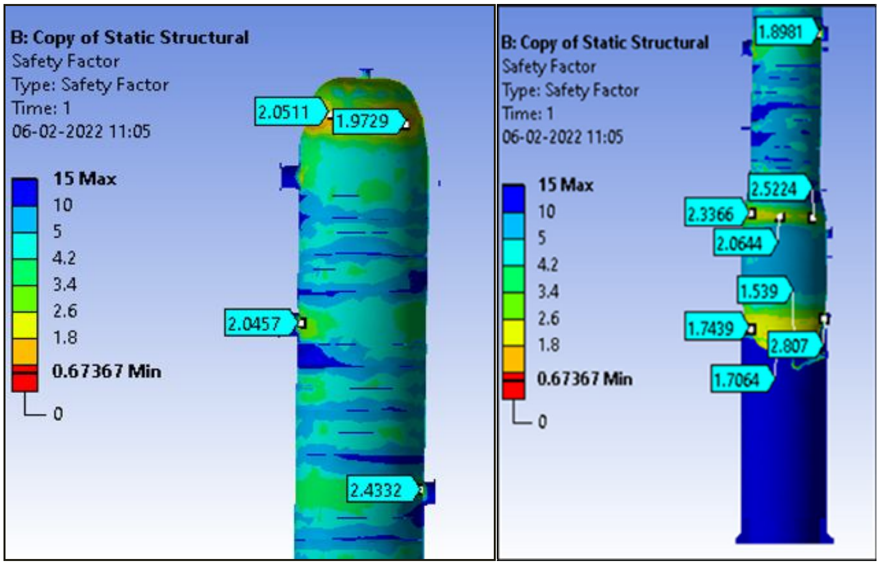


Figure 5. Contour plots for factor of safety

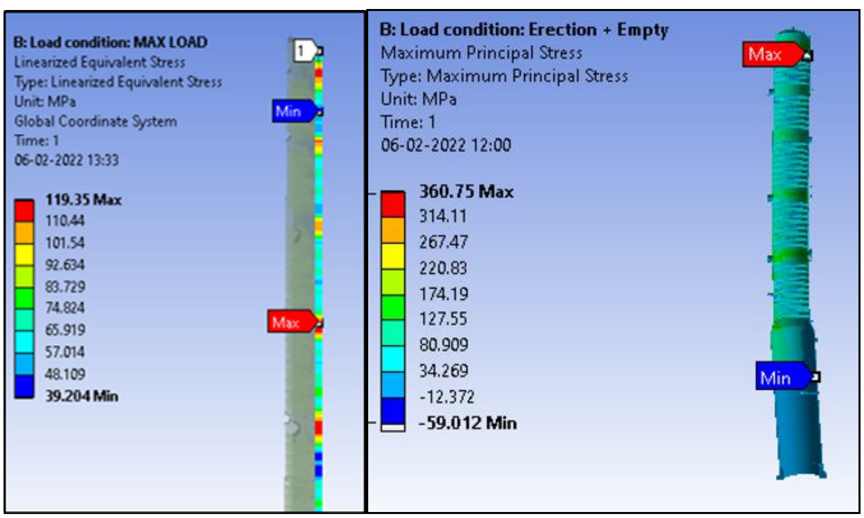


Figure 6. Contour plots foe equivalent stress & maximum principal stress for the tower

In the above case of boundary conditions, the maximum stress will be induced in the third case. So the results in case of the third case are plotted in the figure 6 mentioned below. The equivalent stress & maximum principal stress contours are shown in figure. It is clearly evident that the equivalent stress (342.85 MPa) & the maximum principal stress (360.75 MPa) are well below the maximum tensile strength of the material. So it can be concluded that the design is safe. The total deformation in the vessel is observed at the top of the vessel (82 mm) which is majorly due to the wind loads acting on the vessel. In case of strong winds, the vessel will experience movements at the top of the vessel

FEA results along the stress concentration lines

SCL plot 1: Stress concentration line along the walls of the cylindrical shell at maximum loading condition is shown in figure 6

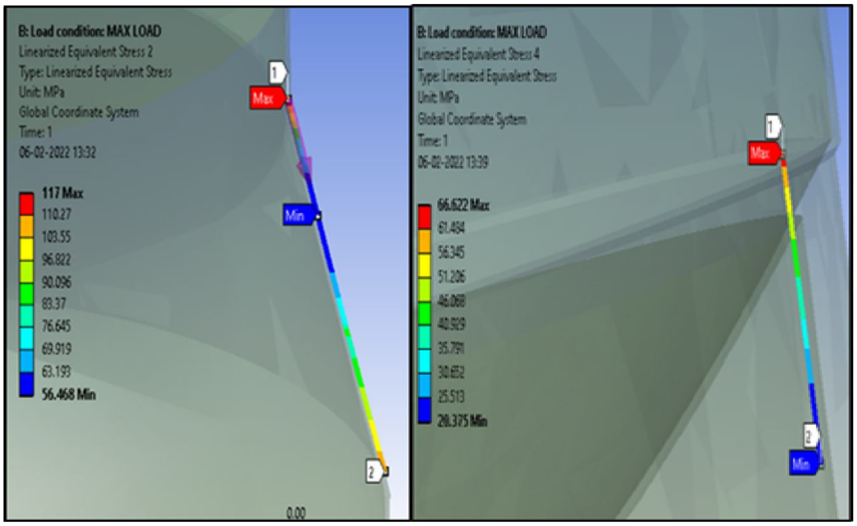


Figure 7. Linearized stresses along the SCL - stress concentration lines

SCL Plot 2: Stress concentration line along the junction between the nozzle & the shell at maximum loading condition is shown in figure 7

SCL Plot 3: SCL at junction of pressure vessel shell & skirt support at maximum load condition is shown in figure 7

VII. CONCLUSION

FEA Analysis has been carried out on the CAD models – Vertical pressure vessel & tall vessel under operating, erection and maximum loading condition & it is clearly visible from the stress plots & contours that the maximum stress induced in the pressure vessel walls in these conditions are well below the permissible allowable stress in the material chosen for manufacturing of the vessel. A tall pressure vessel is designed using ASME Code Section VIII Division – 1 & accordingly FEA has been performed on it using ANSYS to validate the design. The designed pressure vessel is safe for all of the mechanical loads in operating, lifting, erection & operating conditions. The FEA results obtained in the thick cylinder problem matches with the analytical solution & as a result we can conclude that the FEA is successfully executed. Stresses at stress concentration lines are also plotted in the report & areas of high stress concentration are clearly seen in the contours mentioned on the report

REFERENCES

[1] A. M. Senthil Anbazhagan and M. Dev Anand, "FEM with Analytical Approach based Wind and Seismic Design Recommendations for Vertical Tall Process Column," Indian J. Sci. Technol., vol. 9, no. 13, pp. 1–9, 2016, doi: 10.17485/ijst/2016/v9i13/90554

[2] N. A. Rahman, S. R. Sheikh Abdullah, and N. M. Ali, "Interactive short cut method for pressure vessel design based on ASME code," J. Eng. Sci. Technol., vol. 10, no. Spec. Issue 1 on UKM Teaching and Learning Congress 2013, June 2015, pp. 53–60, 2015.

[3] R. Drive, B. Suite, and L. Brundrett, "Pressure Vessel Engineering Ltd. Comparison of Four Head Types Item: Vessel No: Designer: Date: Vessel Name: Reviewed: Michael Tomlinson Four Head Calcs," pp. 1–14, 2016.

[4] N. A. Yahya, O. M. Daas, N. O. F. Alboum, and A. H. Khalile, "Design of Vertical Pressure Vessel Using ASME Codes," no. September, pp. 653–664, 2018, doi: 10.21467/proceedings.4. 33..

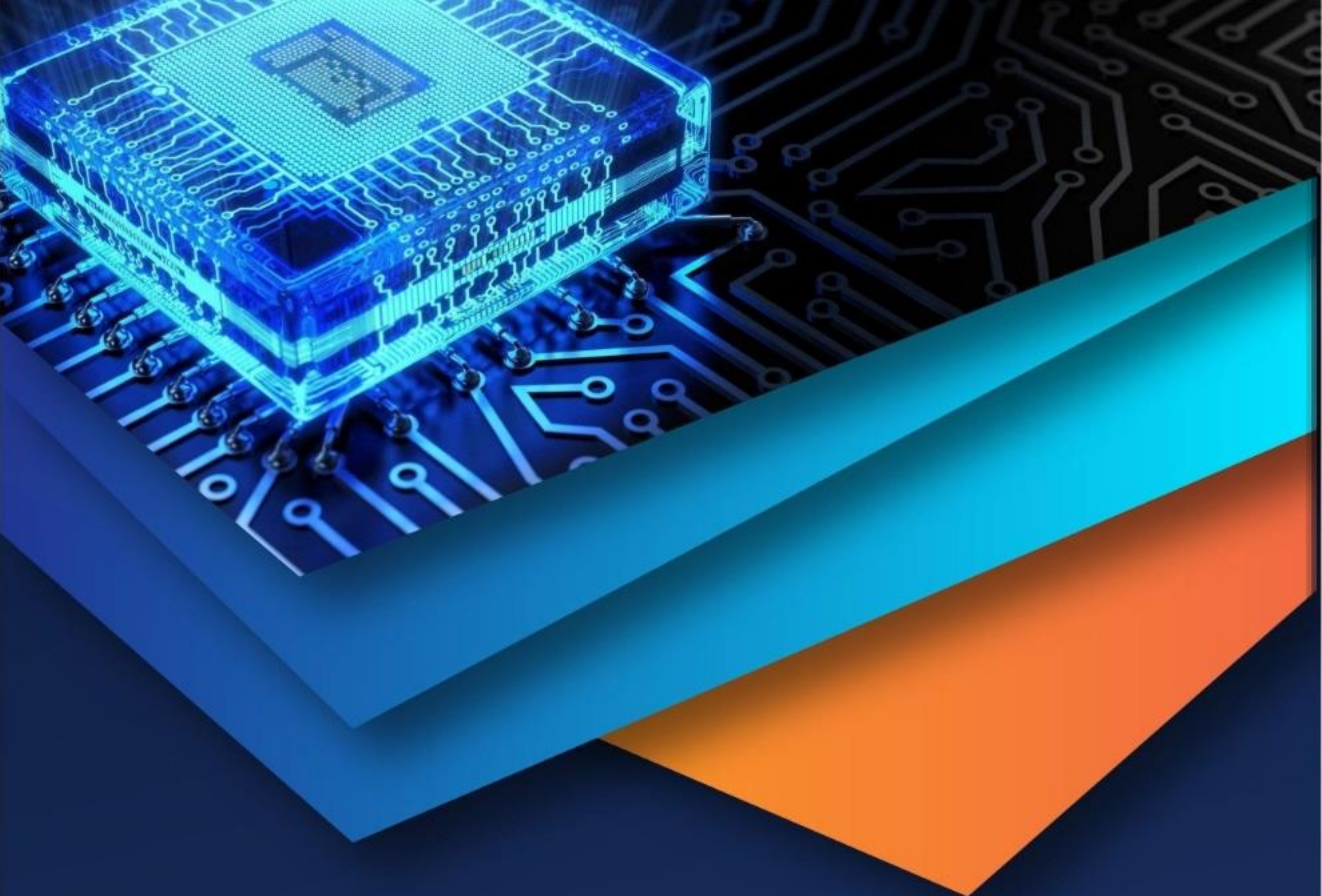
[5] D. K. Jewargi S.S, "Stress Analysis of Pressure Vessel with Different Type of End Connections by FEA," Int. J. Innov. Res. Sci. Eng. Technol., vol. 4, no. 5, pp. 2769–2775, 2015, doi: 10.15680/ijirset.2015.0405016

[6] Dennis Moss, "Pressure vessel design manual"

[7] 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, Issue II, January-March, 2012 A..

[8] ASME Boiler and Pressure Vessel Code 2007 Sec 8 Division 1 (2007).

[9] American Standard Pipe Diameters; http://en.wikipedia.org/wiki/Nominal_Pipe_Size



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

TOGETHER WE REACH THE GOAL