ScienceIn Pubs

Journal of Integrated
SCIENCE& TECHNOLOGY

Analysis of vertical pressure vessel with skirt support

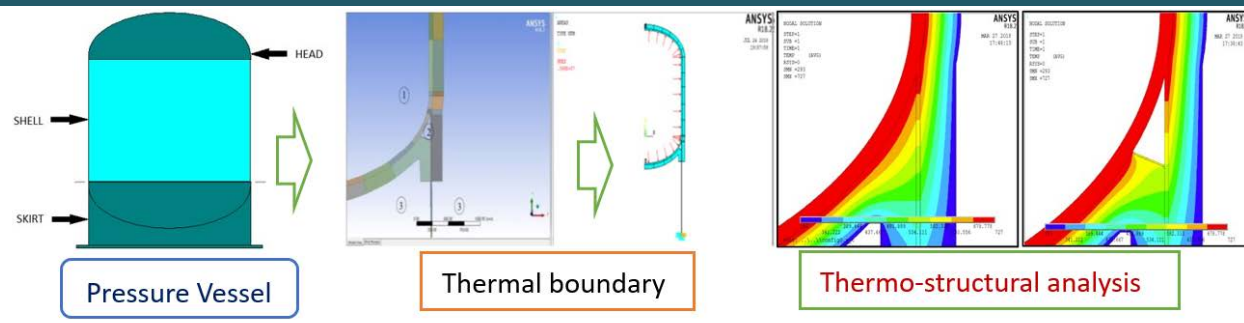
Bharat Murlidhar Shinde[1], Tushar Ankush Jadhav,[2] Omprakash Sugdeo Rajankar,[1] Gopal Malba Alapure,[1] Akash Prakash Choudhari[2]

[1]Dhole Patil College of Engineering, Savitribai Phule Pune University, Pune, India. [2]Sinhagad College of Engineering, Savitribai Phule Pune University, India.

Received on 04-Oct-2023, Accepted on 16 -Dec-2023 and published on: 09-Jan-2024

Article

ABSTRACT



Emphasizing safety has been the top priority in the design of pressure vessels because of their frequent exposure to high-pressure, high-temperature conditions and potential containment of hazardous materials. It is essential to make sure that these containers are impervious to leaks and strong enough to resist the rigors of their operational environment. This study aims to investigate the exact design of a vertical pressure vessel (PV) in accordance to Section VIII, Division 1 of the Boiler and Pressure Vessel (B&PV) Codes of the American Society of Mechanical Engineers (ASME). The study considered various loads by following the UG-22 criteria. Utilizing a 2D axisymmetric model in ANSYS APDL, we conducted a comprehensive simulation to assess the vessel's performance. Following Section VIII, Division 2 of the ASME B&PV Codes, our evaluations cover structural, thermal, and thermo-structural aspects to prevent plastic collapse. Special attention is given to the junction of the vessel's skirt and head, identified as a zone with elevated temperature (Hot Box). This region was subjected to Extensive finite element analysis (FEA) to ensure that the stress levels remain in the acceptable bounds.

Keywords: *Harmonic elements, Pressure vessel, Stress linearization, Thermal analysis, Hot Box, Thermo-structural analysis*

INTRODUCTION

Pressure vessels often operate under conditions of high pressure and temperature, posing significant hazards in accidents, especially when they contain potentially dangerous fluids. Industries such as refineries, chemical plants, and process industries typically employ these vessels, which are usually located away from densely populated areas. However, their potential for catastrophic damage necessitates adherence to established Standard Codes for both design and manufacturing. Regulatory authorities mandate these codes to ensure compliance with safety regulations. Section VIII Division 1 of the ASME[1] B&PV Code, which provides comprehensive instructions for designing vessels in conformance with predefined pressure and temperature parameters, governs the design of pressure vessels. Division 1 specifies design, fabrication, testing, inspection,and certification requirements for PV that operate at internal or exterior pressures greater than 15 psi. Both required and optional appendices contain complementary design criteria, non-destructive testing, and inspection acceptance standards. When creating pressure vessels that operate between 15 and 3000 psi, Division 1 is frequently used. It provides comprehensive formulas for designing each component of the pressure vessel, making it simpler to calculate the necessary thickness, the maximum working pressure, and the levels of stress in the various components. The majority of researchers have conducted finite element analyses on vessels with supports to evaluate their performance and ensure compliance with design requirements. In particular, a finite element analysis on the support

*CorrespondingAuthor: Bharat Murlidhar Shinde, Dhole Patil College of Engineering, Savitribai Phule Pune University, India
Tel: +91-9673995451
Email:bharat_mshinde@yahoo.co.in

Cite as: *J. Integr. Sci. Technol., 2024, 12(4), 792. URN:NBN:sciencein.jist.2024.v12.792*

skirt of a hydrocarbon reactor vessel was done in accordance with Section VIII, Division 2 of the ASME B&PV Code.[1] The bottom head of the vessel, a piece of the cylindrical pressure vessel, and the complete skirt - which includes the base ring, compression ring, and skirt-to-shell junction - were all taken into account throughout the analysis. It is an axisymmetric model since each of these parts was viewed as a shell of revolution. Anchor bolt chairs and other non-axisymmetric components were ignored because they had little bearing on the analysis. Instead of depending on a single point as in a strictly axisymmetric study, stress evaluations were carried out at a minimum of four locations around the circumference (= 0°, 90°, 180°, and 270°) to account for non-axisymmetric mechanical stresses.[2] The stress-strain impact of a high-temperature petrochemical vessel exposed to a hot box has been assessed using ANSYS APDL. A 2D model was constructed using eight-node PLANE 77 elements of the second order. To calculate the thermal stress state, a novel approach was introduced, accounting for radiation as a heat transfer mode within the hot box. Temperature profile data were acquired from solving an axisymmetric problem, revealing a significant temperature gradient increase at the junction of the skirt and shell.[3]

Furthermore, a finite element analysis was conducted to examine the mechanical behavior of the drum-skirt system in the presence of a cracked skirt junction. Measurements were taken for displacement and nondestructive testing of the drum, highlighting the primary risk of plastic collapse in both the skirt and the drum-skirt system. An evaluation of potential mechanical failure risks stemming from skirt junction cracking pointed to the plastic collapse of specific skirt sections due to localized high stress resulting from uneven vertical load distribution on the fractured skirt edge.[4]

Additionally, finite element analysis was employed to investigate stress reduction in cylindrical pressure vessels across three different models. Three key factors significantly influenced stress development in pressure vessels: thickness, nozzle placement, and enclosure head joints. It was observed that increasing vessel thickness led to stress reduction, although this was not considered a viable solution due to cost implications.[5]

Lastly, stress analysis and design optimization of PVs were carried out in accordance with ASME Code, Sec VIII Division 1. Critical points on the PV wall were analyzed using ANSYS, and stress development was assessed for three different materials, ultimately leading to the selection of the most appropriate material.[6]

The PVResearch Council has condensed the three-dimensional stress criteria primarily through the application of elastic two and three-dimensional analytical methods. The primary objective of this work was to provide recommendations for assessing elastic stresses concerning the failure modes defined by the ASME B&PVCode and their correlation with stress limits.[6]

An experimental assessment was carried out to determine strains and temperatures in the hoop and axial directions at the junction area of the skirt and shell. Additionally, finite element analysis (FEM) was conducted using the ANSYS commercial software. Notably, temperature distributions in the shell-to-skirt junction area and the skirt exhibited almost uniformity in the hoop and radial directions. This implies that only temperature gradients in the axial direction were taken into consideration. Consequently, an axisymmetric temperature analysis approach was adopted for the junction area.[7] In this research paper, the objective is to design a PVthat meets specified operating conditions in accordance with ASME B&PV Codes, Section VIII, and Division 1. The primary aim is to determine the minimum required thickness for each component of the vessel. Subsequently, a comprehensive finite element analysis is conducted under various load scenarios. To commence the analysis, a two-dimensional axisymmetric model is constructed using the design modeler within ANSYS Workbench.[8] Structural analysis is performed to identify the maximum equivalent stresses. This is followed by a steady-state thermal analysis to establish the temperature distribution near the junction of the vessel's head and skirt. Furthermore, a Thermo-Structural analysis is executed, incorporating the thermal loads derived from the steady-state thermal analysis. The vessel is subjected to rigorous examination to assess its resistance to plastic collapse under these conditions. Additionally, the study investigates the impact of a hot box on factors such as temperature gradients and thermally induced stresses. Lastly, another static structural analysis is undertaken to assess the vessel's ability to withstand wind and seismic loads, ensuring its structural integrity under these circumstances.

DESIGN CALCULATIONS

We designed the PV using the ASME Code integrated software, PV Elite. We accomplish the design work following the ASME B&PV Codes, Section VIII, Division 1. Instead of designing the components each time, we select them, but this selection is a critical process. According to Section VIII, Division 1 of the ASME B&PV Codes, the vessel is designed taking into account internal pressure, external pressure, the vessel's weight, thermal loads, wind loads, and seismic loads.[9] These considerations are based on the UG-22 standards. Figure 1 depicts the 3-D model of the PVcreated using the graphical software PV Elite. This software allows us to view the PVas a 3-D model while entering the design data. The PV is designed for the operating condition given in Table 1.

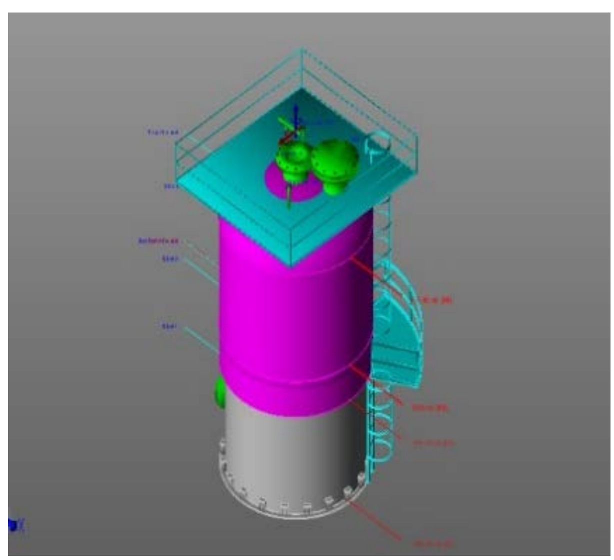


Figure 1 3-D model of PVin PV Elite

Table 1. Operating conditions

Design Parameter	Value
Internal Pressure	3.902 MPa
Internal Temperature	454 ^{0}C
External Pressure	0.017 MPa
External Temperature	20 ^{0}C
Type of Construction	Welded
Corrosion Allowance	3 mm

Table 2. Vessel components

Components	Description	Nominal Thickness (mm)
10 – 20	Skirt 1	18
20 – 30	Skirt 2	18
30 – 40	Bottom Head	74
40 – 50	Shell	66
50 – 60	Top Head	74

The entire vessel is considered to be comprised of the components mentioned in Table 2. All the components are made of SA-387 Gr11 CL2 material except for skirt 1 which is made of SA-516 Gr70.

Thickness Calculations

According to ASME B&PV Code, Section VIII, Division 1, we created the PVcomponents to bear an internal pressure of 3.902 MPa and an external pressure of 0.017 MPa. We determine the minimal thickness needed for the vessel parts and compute the maximum operating pressure as well as the real stresses at the thickness necessary. Then, we contrast these values with the corresponding permissible limits.

Weight of PV

The fabricated weight accounts for the weight of the vessel itself, while the operating weight takes into consideration not only the vessel weight but also includes the weight of the platform, insulation, nozzles, and internals. Specifically, the fabricated weight is 669,708.09 N, whereas the operating weight amounts to 1,578,159.23 N.

Design for wind/seismic loads

We conducted wind load calculations following the guidelines outlined in the Indian Standard IS-875 (Part 3). The initial wind speed considered for this analysis is 170 km/hr. Site-specific data is used to determine the appropriate terrain category and topography factor. Additionally, an equipment class of A is assigned since the vessel's height is less than 20m. Utilizing these parameters, we apply modification factors to calculate the design wind speed (Vz) using equation (1).

$$V_z = Actual\ Speed\ k_1 k_2 k_3 k_4 \qquad (1)$$

where, is the probability factor, *k2* is the roughness of the terrain, *k3* is topography, and *k4* is the importance factor.

The corrected (design) wind speed is employed to compute the shear force and the resulting wind-induced moment. For the evaluation of earthquake-related shear and bending moment values, we utilize the response spectrum method within the PV Elite software. This approach relies on input parameters such as period and acceleration, which are determined based on site-specific data pertaining to the pressure vessel.

Analysis of PV

We examined the PV for plastic collapse failure by assessing the maximum stresses occurring at any point across the vessel's cross-section. These stress values are categorized and then compared to the prescribed allowable limits. The finite element analyses are conducted using a 2D axisymmetric model within the ANSYS APDL software.

Thermal Analysis

We conducted thermal analysis to determine the temperature distribution within the PVwalls. The skirt-to-head junction assumes critical importance in the PVbecause it experiences significant thermal stresses induced by temperature gradients. At this stage, there is a sudden temperature drop along the skirt wall due to a substantial temperature difference. This sharp temperature gradient elevates stresses at the skirt-to-head junction. To mitigate this issue, we introduce an air-pocket or cavity along the skirt wall at the skirt-to-head junction, known as the "hot box." Within the hot box, heat transfer primarily occurs through radiation, facilitating a reduction in the temperature gradient.

We selected temperature-dependent material properties in accordance with ASME B&PV Codes, Section II, Part D. For the thermal analysis, we employ the PLANE77 element, a 2-D 8-node Thermal Solid, with one degree of freedom, temperature, at each node. To apply the radiation boundary condition within the hot box, we utilize the AUX12 Radiation Matrix method in ANSYS APDL.

The boundary conditions for the thermal analysis are divided into three regions, as indicated in Figure 2. In region 1, we maintain the inside surface at the design temperature. In region 2, referred to as the hot box, we apply radiation properties using the radiation matrix method. The emissivity values for surfaces within the hot box are set at 0.8, except for the insulation surface, where an emissivity of 0.7 is utilized. In region 3, we consider ambient conditions, accounting for natural convection. Here, we apply a convective heat transfer coefficient of 25W/m2-K and an ambient temperature of 293K.

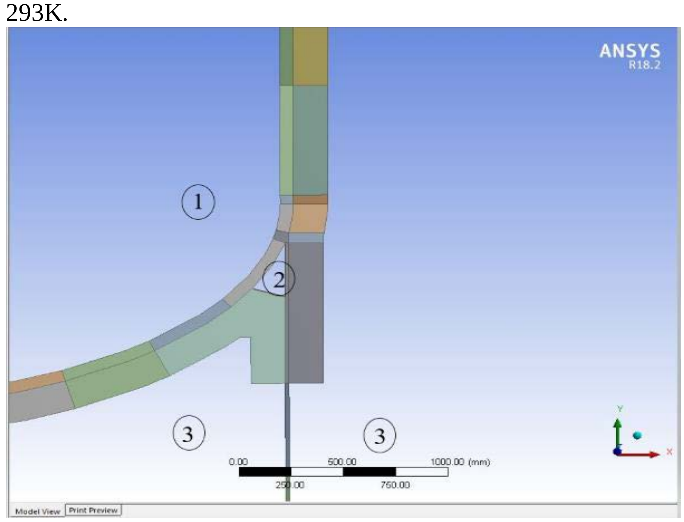


Figure 2 Thermal Boundary Conditions

Thermo-Structural Analysis

In this part of the analysis, we deactivated the insulation, i.e. excluded it from the structural analysis. Instead of relying solely on purely axisymmetric elements, we utilize axisymmetric elements that can accommodate non-axisymmetric loads. Specifically, we employ the PLANE83 element, an 8-node structural solid with axisymmetric-harmonic capabilities. In this analysis, we apply the thermal load in addition to other structural loads, as depicted in Figure 3.

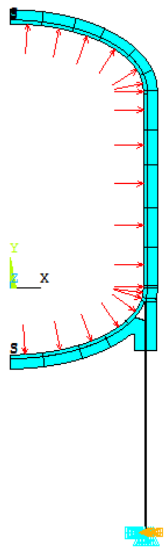


Figure 3 Boundary conditions for thermo-structural analysis

Analysis for wind/seismic loads

While we use an axisymmetric geometry to model the pressure vessel, it's crucial to note that the loads acting on the vessel encompass not only axisymmetric components but also non-axisymmetric forces.[10] Therefore, we employ harmonic elements and select an appropriate Fourier series representation for the applied loads. The research make sure that every part of the vessel is modeled as axisymmetrically as possible during the examination. The fact that each portion naturally symbolizes a shell of revolution makes this possible. The stress profile is split into two sections: the first section deals with axisymmetric loads, while the second section handles non-axisymmetric loads like wind or seismic forces. Harmonic elements play a pivotal role in this analysis as they are uniquely designed to accommodate both axisymmetric and non-axisymmetric loads. By representing a given load in the form of a Fourier series, we express it as a function of the circumferential location (θ), as outlined in equation (2).

$$F(\theta) = \cos(\theta) + B_1 sin(\theta) + A_2 cos(\theta) + B_2 sin(\theta) + \cdots + A_n cos(\theta) + B_n sin(\theta) \qquad (2)$$

Most of the bending loads produced by seismic or wind loads are non-axisymmetric in nature. Harmonic elements and Fourier series load representation are required to include these non-axisymmetric bending loads in the analysis without changing the load distribution around the circumference. Instead of depending on a single point, as is generally done in strictly axisymmetric analyses, stress assessments are carried out at four different positions around the circumference (0°, 90°, 180°, and 270°) due to the presence of non-axisymmetric mechanical loads.

Stress Linearization

To assess resilience against plastic collapse, we conduct an elastic stress analysis. The outcomes of this analysis are sorted and cross-checked against permissible values. Stress Classification Lines (SCLs) are employed, along which stress values are linearized. The adequacy of the vessel for the design conditions is then determined by comparing these stresses to permissible limits. To provide a secure design, we plot SCLs at all crucial locations. Figure 4 shows the locations of the pressure vessel's 2D axisymmetric model's five stress categorization lines.The limits for

$$\left.\begin{array}{l} P_L \angle S_{PL} \\ P_L + P_b \angle S_{PL} \\ P_L + P_b + Q \angle S_{PS} \end{array}\right\} \qquad (3)$$

where,

P_L = Primary membrane equivalent stress

$P_L + P_b$ = Primary membrane plus primary bending equivalent stress

$P_L + P_b + Q$ = Primary membrane stress plus primary bending stress plus secondary stress

respective stresses are as follows

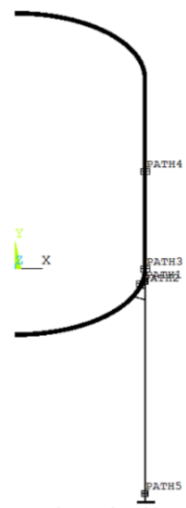


Figure 4 Stress Classification Lines

Also, the allowable stress values are as follows;
Allowable stress (S)= 141.68 MPa ;
Yield stress (Sy) =310 MPa ;
SPL = Sy = 310 MPa;
SPS = 2Sy = 620 MPa

Results and Discussion

Analytical Results

According to the design calculations, Table 3 provides the minimum required thickness and the chosen design thickness for each component. Observation reveals that the nominal thickness values exceed the minimum required thickness values, even accounting for a corrosion allowance of 3mm.[4]

Table 3. Thickness results for vessel components

Component	Description	Nominal Thickness (mm)	Required Thickness (mm)
10 – 20	Skirt 1	18	8.74
20 – 30	Skirt 2	18	8.74
30 – 40	Bottom Head	74	63.33
40 – 50	Shell	66	64.30
50 – 60	Top Head	74	63.33

FEA Validation 245

By employing the ASME Code-integrated software PV Elite, we determined the circumferential stress, which was measured at 133.49 MPa. In contrast, utilizing finite element analysis (FEA), the circumferential stress was determined to be 132.79 MPa, as depicted in Figure 5. This comparison aids in confirming the adequacy of vessel thickness for each component.

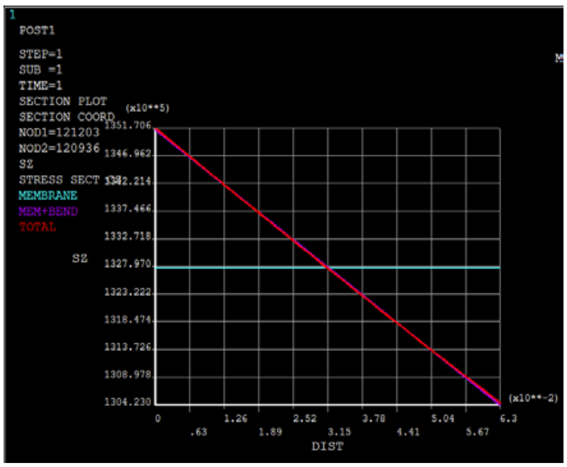


Figure 5 linearized stresses in z-direction

Table 4. Circumferential stress result

Stress	Analytical result	FEA Result	Limit
Circumferential Stress (MPa)	133.49	132.79	141.68

The analytically calculated circumferential stress closely aligns with the FEA-derived value, exhibiting a negligible error of just 0.5%. Moreover, it falls comfortably within acceptable limits.[11]

Effect of Hot Box

To check the reduction in temperature gradient in the presence of a hot box, we conducted thermal analysis for two different configurations. One configuration included a hot box, while the other did not.[3] We measured the temperature readings along the skirt wall at a distance of 30 cm from the skirt to the head junction and calculated the temperature gradient for both cases. The thermal analysis plots for both cases are presented in Figures 6 and 7.

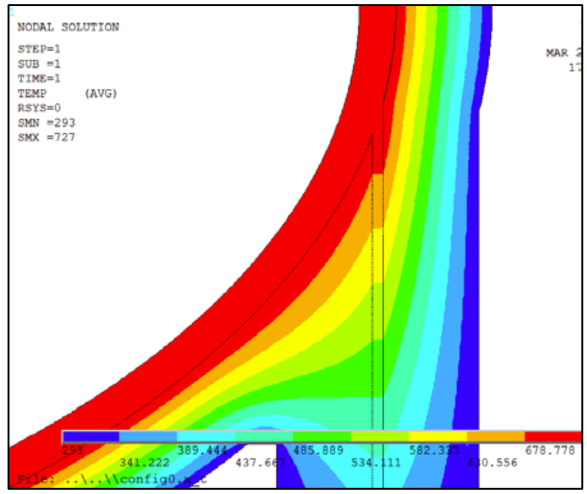


Figure 6 Temperature distributions without hotbox

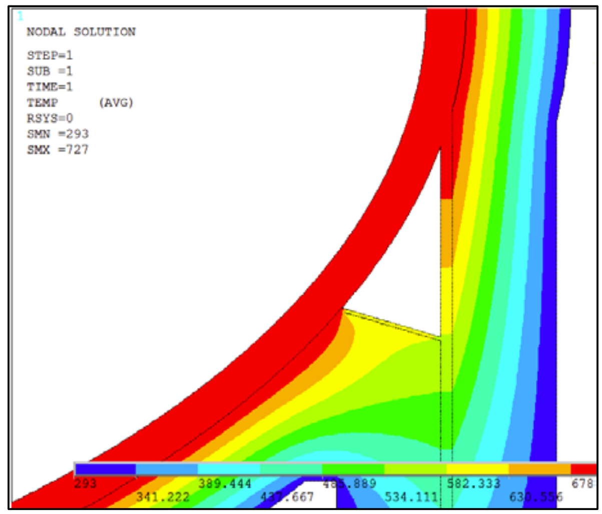


Figure 7 Temperature distributions with hotbox

Figures 6 and 7 show that the length of each temperature zone is slightly increased when a skirt with a hot box is used. This indicates a more gradual reduction in temperature along the skirt wall compared to the configuration without a hot box. Therefore, we recommend using a hot box in cases with high external loadings and extreme operating conditions. The temperature distribution can be better understood through temperature versus distance plots. Notably, at the same 30 cm distance, there is a temperature difference of 22.24 K.

Figures 8 and 9 reveal that the presence of a hot box reduces the thermal gradient from 0.52 K/mm to 0.45 K/mm, resulting in a decrease in thermal stress. We performed a thermostructural analysis to confirm this, applying the temperature distribution from the thermal analysis as thermal loads.The results of the thermo-structural analysis were used to assess the impact of the hot box by comparing stress values with and without it, as shown in Tables 5 and 6.

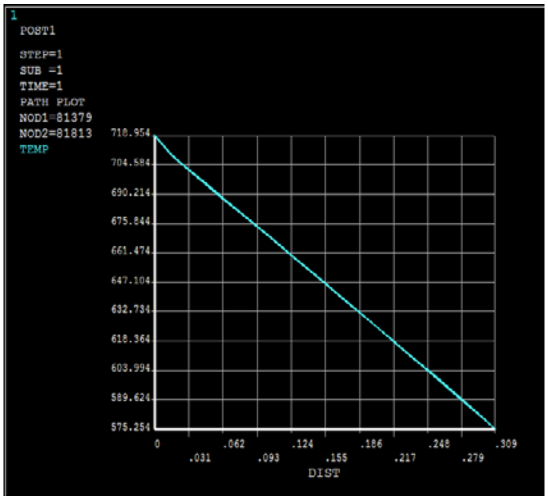


Figure 8 Temperature plot with hot box

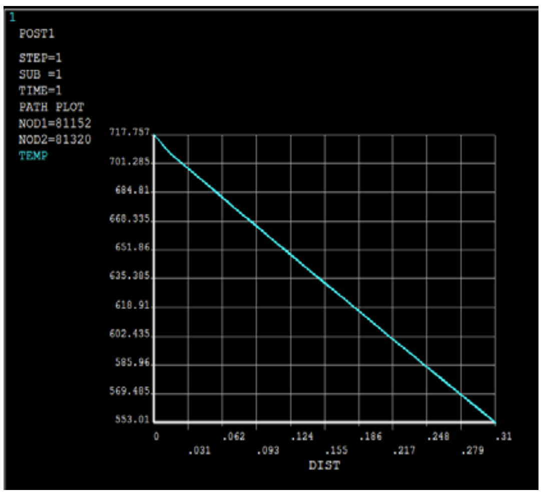


Figure 9 Temperature plot without hot box

Tables 5 and 6 clearly demonstrate a significant reduction in stresses near the skirt-to-head junction due to the presence of the hot box. This reduction is evident in the stress values obtained at Path1 and Path2. In Path 2, we observed a 6% reduction in membrane stress, which is considered more critical and warrants greater attention during stress analysis. These reduced stress values indicate the positive impact of incorporating a hot box, resulting in a reduction of thermally induced stresses.

Table 5. Stress values without hot box

SCL	P_L (MPa)	$P_L + P_b + Q$ (MPa)
Path1	39.98	251.1
Path2	121.4	206.7
Path3	63.36	82.28
Path4	116.5	119.5
Path5	5.96	6.29
Allowable stress	**310**	**620**

Table 6. Stress values with hot box

SCL	P_L (MPa)	$P_L + P_b + Q$ (MPa)
Path1	43.48	201.2
Path2	114	202.9
Path3	63.06	82.83
Path4	116.5	119.5
Path5	5.76	6.56
Allowable stress	**310**	**620**

CONCLUSIONS

In conclusion, our design and analysis of the PVadhere meticulously to ASME Codes, ensuring structural integrity and safety. We determine the minimum required thickness for all vessel components and validate these values by assessing the maximum allowable working pressures and stresses in each component. The finite element analyses, conducted in line with ASME B&PV Codes, Section VIII, Division 2, demonstrate close alignment between analytically calculated circumferential stress and FEA results, with a minimal error of 0.5%, well within acceptable limits. The introduction of a hot box significantly reduces the thermal gradient, resulting in lower thermal stresses. Thermo-structural analysis reveals a 6% reduction in membrane stresses near the junction along Path2. Stresses are linearized and assessed for plastic collapse, consistently falling well within the allowable limits prescribed by ASME codes. Stress Classification Lines (SCL) are instrumental in this evaluation, allowing us to analyze the total stress distribution on a component basis. The reaearch measure stresses at four sites around the circle (0°, 90°, 180°, and 270°), surpassing the single-location approach customary in pure axisymmetric calculations due to the non-axisymmetric nature of seismic loads.We also discuss the application of harmonic elements and Fourier series to incorporate non-axisymmetric seismic loads into the 2-D axisymmetric model of the pressure vessel.

In sum, our comprehensive analysis and design ensure the pressure vessel's robustness against various loadings, as stipulated in UG-22 of ASME B&PV Codes Section VIII Division 1, affirming its safety and reliability.

CONFLICT OF INTEREST

The authors declared no conflict of interest the for publication of this work.

REFERENCES AND NOTES

1. J.W. Stokes, L. Holly, K.H. Mayer. The ASME boiler and pressure vessel code. *Non-Destructive Test.* **1974**, 7 (3), 145–151.
2. D.K. Williams, T.G. Seipp. Considerations in the design and analysis of an ASME section VIII, Div. 2 reactor support skirt. *J. Press. Vessel Technol. Trans. ASME* **2007**, 129 (2), 316–322.
3. I. Laskin, B. Volfson, P. Redikultsev. The study of the effect of the hot box on the stress-strain state of the high temperature petrochemical vessels. *ASME 2018 Symposium on Elevated Temperature Application of Materials for Fossil, Nuclear, and Petrochemical Industries, ETAM 2018.* 2018.
4. J.L. González, S. Gómez, G. Gómez. Analysis of the mechanical behavior of a delayed coker drum with a circumferentially cracked skirt. *Procedia Struct. Integr.* **2017**, 3, 33–40.

5. F. Nabhani, T. Ladoku. Reduction of Stresses in Cylindrical Pressure Vessels. In *Finite Element Analysis - From Biomedical Applications to Industrial Developments*; **2012**.
6. G. Hollinger, J. Hechmer. Three-dimensional stress criteria - summary of the PVRC project. *J. Press. Vessel Technol. Trans. ASME* **2000**, 122 (1), 105–109.
7. M. Oka, H. Ambarita, M. Daimaruya, H. Fujiki. Study on the effects of switching temperature on the thermal fatigue life of the shell-to-skirt junction of coke drum. *J. Press. Vessel Technol. Trans. ASME* **2011**, 133 (6), 61210.
8. B. Siva Kumar, P. Prasanna, J. Sushma, K.P. Srikanth. Stress Analysis and Design Optimization of A Pressure Vessel Using Ansys Package. *Mater. Today Proc.* **2018**, 5 (2), 4551–4562.
9. D.K. Williams, T.G. Seipp. Considerations in the Design and Analysis of an ASME Section VIII, Div. 2 Reactor Support Skirt. *J. Press. Vessel Technol.* **2007**, 129 (2), 316–322.
10. C.E. Freese. Vibrations of Vertical Pressure Vessels. *J. Eng. Ind.* **1959**, 81 (1), 77–86.
11. T.P. Pastor, J. Hechmer. Asme task group report on primary stress. *J. Press. Vessel Technol. Trans. ASME* **1997**, 119 (1), 61–67.