Design and Failure Analysis of a Vacuum Pressure Vessel for Aerospace Applications using Finite Element Analysis (FEA)

Eiman Solangi
Department of Mechanical Engineering, University Technology Petronas, Seri Iskandar, Malaysia
eiman_22010932@utp.edu.my (corresponding author)

Thar Muhammad Badri Albarody
Department of Mechanical Engineering, University Technology Petronas, Seri Iskandar, Malaysia
dher.albarody@utp.edu.my

Sarah Al-Challabi
Department of Mechanical Engineering, University Technology Petronas, Seri Iskandar, Malaysia
sarah_21001298@utp.edu.my

Javed Akbar Khan
Department of Electrical and Mechanical Engineering, China University of Petroleum, Qingdao, China
javedakbar.khan@upc.edu.cn

Sajjad Ali
Department of Mechanical Engineering, University Technology Petronas, Seri Iskandar, Malaysia
sajjad_22010080@utp.edu.my

Received: 29 April 2024 | Revised: 19 May 2024 | Accepted: 22 May 2024

ABSTRACT

This paper provides detailed insights into the external pressure vessels. External pressure vessels are designed to perform under extreme operating conditions, so the selection of material and geometry along with stress analysis are fundamental for their optimal efficiency. The objective of this research is to provide a case study of the material, design, and stress generated in the pressure vessel to make it suitable for thermal and load-bearing applications in the aerospace industry. An aluminum alloy was chosen as the design material due to its low density and high strength. The modeling geometry of the alloy was constructed using the ASME section-VIII division-I div code. After the performed Finite Element Analysis (FEA), modeling was carried out deploying the ANSYS design modeler to obtain the stress concentration and failure mode of the model. The present study demonstrates the behavior of a structure under applied load and identifies the weak areas of its geometry. Based on the external stress at the center of the structure, the maximum and minimum stresses computed are 0.763 MPa and 0.00803 MPa, respectively. It was also found that the maximum strain is generated at the center of the structure and is equal to 1.0834e-6 mm/mm, the maximum deformation is equal to 0.00109 mm and also occurred at the center, while the shells of the model are unable to undergo any deformation. FEA results agree with the analytical results, as the errors for hoop stress and equivalent strain are 4.8% and 1.8%, respectively. Thus, the proposed method can be applied to predict the equivalent stress, equivalent strain, total deformation, and stress intensity, which are required for the structural integrity analysis of pressure vessels.

Keywords-finite element analysis; pressure vessel; ASME code compliance; stress concentration

I. INTRODUCTION

Pressure vessels are sealed containers designed to retain the fluids at a pressure considerably higher or lower than that of the ambient pressure [1]. These vessels' applications are detected in various industries, such as the aerospace, petrochemical, oil, and gas. They are also utilized in chemical and heat treatment and food processing industries [2]. Over the past few decades, a large number of industrial sectors have implemented lightweight structures because they are more resilient, flexible, easily fabricable, and have a lower operating and manufacturing cost. Pressure vessels have sparked huge interest in scientists and designers in the aerospace sector because they are manufactured with materials which have lower density. Previously, lifting gases, such as hydrogen and helium, were used to generate lift, but hydrogen is extremely combustible. Both helium and hydrogen are potent greenhouse gases, and are thus now being replaced by vacuum. These structures provide sustainable space exploration and could potentially be the future of aerospace. When the pressure vessel is exposed to this type of pressure, the material of the vessel is subjected to pressure loading, and hence to stresses, from all directions [3]. Authors in [4, 5] indicated that the pressure vessels, based on their different models and applications, can be carefully designed to eliminate or minimize any potential hazard. Depending upon their application, the design of the pressure vessels may be multifaceted due to complex geometry, combined thermal loads, structural stresses, and aggressively harsh environment. In such conditions, it is often difficult for the analytical model to be analyzed. It is therefore necessary to have a complete design and stress analysis of such a model to avoid operational failure [6].

Cost is an important factor to be considered while designing and fabricating a pressure vessel. Authors in [7] conducted a low-cost validation on a thin-walled pressure vessel. They conducted FEA, using distortion energy as a validation tool for safety, and concluded that the distortion energy theory cannot ensure model safety. Moreover, the cost analysis demonstrated that the failure theory can save some cost in pressure testing. The primordial method deployed for obtaining stress concentration in a design ensures the overall stability of the pressure vessel. Authors in [8] studied the recent advancements in determining the stress concentration factor in pressure vessels and they documented that a high localized bending stress does exist in the structure. All the other applied forces and loading conditions of the pressure vessels should be thoroughly considered for better performance. The model of the pressure vessel can be optimized by slightly changing its geometry or through material enhancement. Authors in [9] conducted a stress analysis of a pressure vessel performing FEA and putting forward an optimized model with improved stress development in critical locations. Authors in [10] analyzed vertical pressure vessels manufactured by utilizing the ASME code after performing FEA. Employing 2D quad meshing, they determined the fatigue life of the structure and concluded with a theoretical validation of the entire model, with the results obtained being under the desired limit. Other factors, such as displacement and rotational velocity, can also affect the overall working efficiency of a pressure vessel. Authors in [11] investigated the effect of rotational velocity on the performance of a pressure vessel. They carried out an FEA of the structure and validated the results by comparing them with the results obtained from a manually computed method. Their investigation showed that the model is safe within the particular working conditions. However, the findings acquired from the FEA were more accurate than those obtained manually. Authors in [12] conducted thermal analysis of a composite material and a comparative analysis for structural steel and aluminum alloy, performing FEA. The model was designed by ASME configurations and the results disclosed that the model fabricated by structural steels exhibited better stress tolerance due to the high yielding strength of the material. Authors in [13] carried out stress analysis on a traditional company's designed equipment and subsequently conducted FEA to improve the deficiencies of the traditional model. The results revealed that the stress value of the equipment was accurately calculated and was compared with the standard design. Also, the designed equipment rationality was determined, with the efficiency having been significantly improved.

II. RESEARCH OBJECTIVES

The main concern regarding external pressure vessels is their structural integrity. According to [14], the model needs to be manufactured from materials with a high mechanical stress tolerance to maintain its structure under highly hard operating circumstances. These external pressure vessels allow sustainable space exploration and can be used for surveillance and cargo purposes [15]. Designers, though, have not put forward yet a stable vacuum pressure vessel for aerospace application despite the years of advancement in material sciences and manufacturing processes. This research contributes to the analysis of such a model, which can potentially be the future of aerospace. In this paper, FEA using ANSYS is performed to achieve this objective. The geometry is created utilizing the design modeler, following the ASME design code for pressure vessels. Afterwards, the model is subjected to boundary and loading conditions, before finalizing the results, with a mesh independence study being conducted to achieve result accuracy. Finally, FEA is performed to determine the stress concentration in the model by identifying the most vulnerable areas in the geometry to minimize the stress effect and increase structure viability. The results obtained from this analysis refer to a total deformation in the geometry, a maximum equivalent (von Mises) stress, strain generated in the model due to the applied stress, and finally, stress intensity.

III. METHODOLOGY

Figure 1 depicts the methodology followed.

Fig. 1. Process flow of research methodology.

A. Design Considerations

1) Material Selection

The selection of the materials is the most crucial aspect when it comes to the design of any model. The main criterion for selecting suitable materials is the potential applications of the model. Pressure vessels have extremely harsh operating conditions, so the materials they are made of must have high yielding strength to withstand the operating conditions [15, 16]. Carbon steel, carbon fiber, aluminum, and titanium alloys with high tensile strength are usually preferred for such designs. The present study considered aluminum alloy AA6061 for the pressure vessel manufacturing, due to its remarkable mechanical properties (Table I), such as lower density and higher hardness, which are perfectly suitable for load bearing structures. AA6061 pressure vessels are ideal as they are lightweight, safe, and have high yield strength. In AA6061 alloy, aluminum is the predominant metal, while the remaining constituents are silicon, copper, and magnesium.

TABLE I. MECHANICAL PROPERTIES OF AA6061

Property	Symbol	Unit	Value
Density	ρ	kg/m^3	2660
Elastic modulus	E	MPa	70,300
Poisson's ratio	υ	-	0.3
Yield stress	δy	MPa	125
Ultimate stress	δu	MPa	275
Design stress	δd	MPa	93.75

2) Pressure

The design pressure must be carefully evaluated for the smooth operation of the pressure vessels. The internal and external pressure of the vessel must, respectively, be higher and lower from the ambient pressure [16]. In the case of the vacuum pressure vessel, the internal pressure must be lower than the outside atmospheric pressure, 14.7 psi.

3) Temperature

The temperature needs to be substantially controlled because the pressure and mechanical properties of the materials depend on temperature, as some materials have a ductile nature at high temperature and a brittle one at room temperature. So, the pressure vessel must operate at a temperature lower than the evaluated allowable stress value [17].

4) Allowable Stress

The maximum allowable stress values for pressure vessels are determined from ASME. Other important parameters for the construction of highly efficient pressure vessels are the precise control of corrosion, joint efficiency, maintenance, and quality control [18].

B. Finite Element Analysis

FEA is a computational technique, which helps determine the complete structural analysis of a model involving its complex geometry [19]. This numerical analysis must be performed beforehand to determine the stress concentration on the structure's geometry and find the regions where failure is expected to occur.

1) Pre-Processing

Pre-processing is the initial FEA stage, where geometry is modeled or imported from different designers. In this research the geometry of the pressure vessels is generated in an ANSYS design modeler using the ASME design code. Figure 2 illustrates the model, which is built according to the dimensions given in a previous study [20].

TABLE II. 3D MODEL DIMENSIONS

Dimensions	
Shell's thickness	6.35 mm
Diameter	203.2 mm
Radius	101.6 mm
Shell's length	394 mm

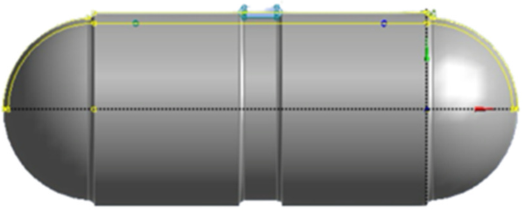

Fig. 2. FEA geometry.

2) Boundary Conditions

The geometry is assigned a material, whose properties are listed in Table I. Then, loading and boundary conditions are given to the 3D model, as evidenced in Table III. To generate partial or full vacuum in a pressure vessel, the internal pressure must be subsequently lower than the pressure outside the shell, as shown in Figure 3.

TABLE III. BOUNDARY AND LOADING CONDITIONS OF THE MODEL

Boundary and loading conditions	
Internal pressure	7.25 psi
External pressure	14.7 psi
Temperature	22 °C
Fixed support	Head of shell
Mesh size	10 mm

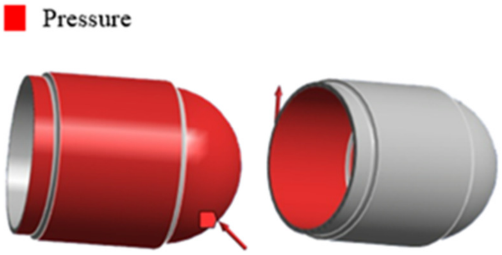

Fig. 3. Internal and external pressure applied on the model.

C. Model Validation

The FEA results were validated by the analytical formula of stress on the basis of the design code for ASME pressure vessels [20]:

$$\sigma_{\text{hoop}} = \frac{P \times R}{t \times E - 0.6 \times P} \tag{1}$$

$$\sigma_{\text{hoop}} = \frac{0.05 \times 101.6}{6.35 \times 1 - 0.6 \times 0.05} \quad \sigma_{\text{hoop}} = 0.803\text{MPa}$$

where P is the internal pressure, R is the internal radius, t is the thickness of shell and E is the joint efficiency.

Hoop stresses are generated in the model when pressure is applied along the circumference of the cylinder, and they can be calculated based on the formulae from the ASME design code. Elastic strain is calculated by Hook's law:

$$\sigma = \epsilon \times \mathrm{E} \qquad (2)$$

$$\epsilon = \frac{\sigma}{\mathrm{E}} \qquad \epsilon = \frac{0.803}{73{,}000} \qquad \epsilon = 1.1\mathrm{e}^{-5}\mathrm{mm/mm}$$

where E is the elastic modulus.

IV. RESULTS

After designing the geometry and applying boundary and loading conditions, FEA was conducted and the results concerning total deformation, equivalent stress, equivalent strain, and stress intensity were obtained. Prior to conducting FEA, mesh sensitivity testing was performed to ensure that the results converge to a constituent solution. Moreover, for the error to be minimized, it had to be ensured that the results were invariant with respect to meshing. Three types of mesh styles were deployed, coarse mesh, medium mesh, and fine mesh. Initially, the simulations were conducted utilizing a coarse mesh with an element size of 25 mm. Afterwards, the same process was performed using a medium mesh, and lastly fine mesh was employed, with a reduced element size, changing none of the nodes and none of the elements in each iteration.

A. *Equivalent Stress (Von-Mises) and Elastic Strain*

Figure 4 presents the equivalent stress, demonstrating the behavior of the structure under applied load and identifying the weak areas geometry [21]. In this model, the maximum stress is applied at the center of the geometry and the minimum at the shell. Their values are 0.763 and 0.00803 MPa, respectively. Figure 5 displays the strain on the model due to the applied maximum stress. The maximum strain is generated at the center of the structure and equals to 1.0834e^{-6} mm/mm.

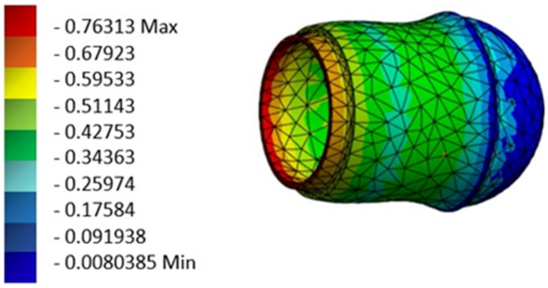

Fig. 4. Equivalent von-Mises stress distribution.

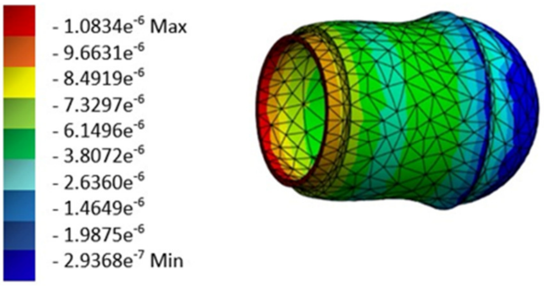

Fig. 5. Equivalent strain.

B. *Total Deformation and Stress Intensity*

Figure 6 showcases the total deformation of the model, exhibiting the areas of the geometrical vessel which are most affected by the applied forces [22]. In this model, a maximum deformation of 0.00109 mm occurred at the center, whereas the shells of the model did not undergo any deformation. Figure 7 depicts the intensity of the stress in the model, which is 0.0877 MPa. The results obtained from the FEA can be seen in Table IV. Subsequently, these results were validated through numerical analysis based on the ASME design code. The contrast in the results is outlined in Table V, along with the error percentage estimation. The results obtained from the FEA were in close correlation with the analytical result data. However, the maximum stress attained from the FEA is more accurate and relatively reduced compared to the stress value derived from the analytical method. These results also identify the most vulnerable areas in the geometry, which are located to the center part of the model.

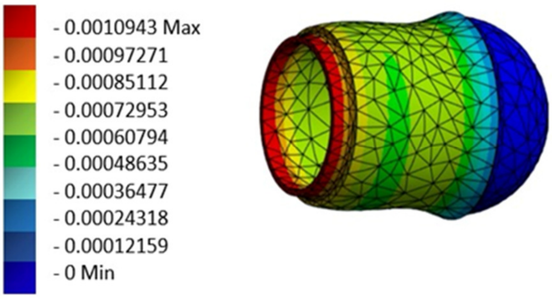

Fig. 6. Total deformation.

TABLE IV. FEA RESULTS

FEA result evaluation		
	Maximum value	**Minimum value**
Equivalent (von Vises) stress	0.763 MPa	0.0080MPa
Equivalent elastic strain	1.083e^{-5} mm/mm	2.97e^{-7} mm/mm
Total deformation	0.00109 mm	0 mm
Stress intensity	0.807 MPa	0.0091 MPa

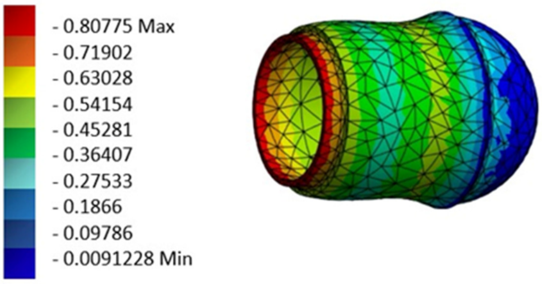

Fig. 7. Stress intensity.

TABLE V. ANALYTICAL AND FEA RESULT COMPARISON

	Analytical method	**FEA**	**Error %**
Hoop stress δ	0.803 MPa	0.763 MPa	4.8%
Equivalent strain ϵ	1.08e^{-5} mm/mm	1.1e^{-5} mm/mm	1.8%

V. CONCLUSIONS

This paper presents a comprehensive case study on the design and stress analysis of vacuum pressure vessels. The analysis started with the material selection process by reviewing their mechanical and chemical properties. Then, a 3D model was designed and validated based on the ASME standard code. Boundary and loading conditions were subsequently applied to the model. Finally, FEA was conducted using ANSYS 2023. Before obtaining the results, mesh independence analysis was performed to ensure that the error was minimized, with the results being invariant with respect to the mesh setting. A fine mesh with 10 mm of mesh element size was selected, as it provides the optimal results.

The present study provides a new procedure for evaluating the structural integrity of pressure vessels. The results obtained from the FEA constituted a total deformation in the model, an equivalent (von-Vises) stress developed in the critical areas, and strain generated in the geometry due to applied stress. The findings are in agreement with the literature [21]. In the proposed model, the maximum stress, which equals to 0.763 MPa, is applied at the center of the geometry, whereas the minimum stress, which equals to 0.00803 MPa, is induced in the shell. Moreover, the maximum strain is generated at the center of the structure and equals to $1.0834e^{-6}$ mm/mm. The total deformation in the presented model shows a similar trend to that evidenced in [22], demonstrating the geometry areas which are most affected by the forces applied. A maximum deformation of 0.00109 mm occurred at the center, whereas the shells of the model did not undergo any deformation. In conclusion, it was discovered that the center of the model is the most vulnerable area, as the maximum stress is concentrated around it, causing maximum deformation. The outcome was then contrasted with the ASME design analysis. The results obtained from the performed FEA were compared and validated with the analytical model of the stress according to the design code for ASME pressure vessels. The results attained from the finite element method are in close agreement with those of the analytical method, as the errors for hoop stress and equivalent strain are 4.8% and 1.8%, respectively.

VI. FUTURE RECOMMENDATIONS

- Enhancement in the engineering materials and manufacturing processes will always have room for improvement despite the advancements made in this field. The latest periodic cellular structures of materials allow designers to fabricate materials with the desired mechanical properties.
- The geometry of the pressure vessels can be altered for the even distribution of stresses generated in complex locations to be achieved, thus improving their overall efficiency.
- Advanced monitoring and predictive maintenance should be set up to optimize vessel performance and reduce downtime by detecting premature stress distortion, which leads to permanent catastrophic failure.

ACKNOWLEDGMENT

The authors would like to extend their acknowledgement to the University Technology PETRONAS, for its financial support, under the grants (MRA 015MDO-151) and (015BD1-001), for which it is gratefully appreciated.

REFERENCES

[1] K. Khadke and Dr. D. Chawde, "Design & FEA of Pressure Vessel," *International Journal for Research in Applied Science & Engineering Technology (IJRASET)*, vol. 10, no. 7, pp. 4741–4748, Jul. 2022, https://doi.org/10.22214/ijraset.2022.46076.

[2] D. Koulocheris and C. Vossou, "Exploration of Equivalent Design Approaches for Tanks Transporting Flammable Liquids," *Computation*, vol. 8, no. 2, Jun. 2020, Art. no. 33, https://doi.org/10.3390/computation8020033.

[3] S. Moyya, P. Thejasree, B. Cherian Abraham, and G. S. Mangalathu, "Design and analysis of single and multi-layer pressure vessel," *Materials Today: Proceedings*, Jul. 2023, https://doi.org/10.1016/j.matpr.2023.06.393.

[4] V. Chowdary, "Design and Static Analysis of Different Pressure Vessels and Materials Using FEM Method," *International Journal of Advance Scientific Researchand Engineering Trends*, vol. 5, no. 7, pp. 93–99, Jul. 2020.

[5] I. Gophane, N. Dharashivkar, P. Mulik, and P. Patil, "Theoretical and FEA of Pressure Vessel," *Indian Journal of Science and Technology*, vol. 17, no. 12, pp. 1148–1158, Mar. 2024, https://doi.org/10.17485/IJST/v17i12.3272.

[6] I. Ramos, Y. Ho Park, and J. Ulibarri-Sanchez, "Analytical and Numerical Studies of a Thick Anisotropic Multilayered Fiber-Reinforced Composite Pressure Vessel," *Journal of Pressure Vessel Technology*, vol. 141, Dec. 2018, Art. no. 011203, https://doi.org/10.1115/1.4041887.

[7] J. Pratama, D. F. Fitriyana, Rusiyanto, J. P. Siregar, and W. Caesarendra, "A low cost validation method of FEA on a thin walled vertical pressure vessels," *Journal of Physics: Conference Series*, vol. 1444, no. 1, Jan. 2020, Art. no. 012042, https://doi.org/10.1088/1742-6596/1444/1/012042.

[8] G. Terán, S. Capula-Colindres, J. C. Velázquez, D. Angeles-Herrera, and E. Torres-Santillán, "On the Influence of the Corrosion Defect Size in the Welding Bead, Heat-Affected Zone, and Base Metal in Pipeline Failure Pressure Estimation: A FEA Study," *Journal of Pressure Vessel Technology*, vol. 141, Apr. 2019, Art. no. 031001, https://doi.org/10.1115/1.4042908.

[9] B. Siva kumar, P. Prasanna, J. Sushma, and K. P. Srikanth, "Stress Analysis And Design Optimization Of A Pressure Vessel Using Ansys Package," *Materials Today: Proceedings*, vol. 5, no. 2, Part 1, pp. 4551–4562, Jan. 2018, https://doi.org/10.1016/j.matpr.2017.12.025.

[10] S. J. Niranjana, S. V. Patel, and A. K. Dubey, "Design and Analysis of Vertical Pressure Vessel using ASME Code and FEA Technique," *IOP Conference Series: Materials Science and Engineering*, vol. 376, no. 1, Mar. 2018, Art. no. 012135, https://doi.org/10.1088/1757-899X/376/1/012135.

[11] M. M. Islam, M. F. Hossain, M. S. Rana, and M. S. Ferdous, "Burst pressure prediction of carbon/natural fiber and hybrid composite pressure vessels by theoretical and numerical analysis," *Hybrid Advances*, vol. 5, Apr. 2024, Art. no. 100172, https://doi.org/10.1016/j.hybadv.2024.100172.

[12] A. Kothali, U. Bhapkar, and J. Bhat, "FEA of bursting pressure in FRP pressure vessel," *Materials Today: Proceedings*, vol. 56, pp. 2932–2937, Jan. 2022, https://doi.org/10.1016/j.matpr.2021.10.358.

[13] J. Sun, J. Xin, and Y. Wang, "Stress Analysis of Pressure Vessels Based on ANSYS," *International Journal of Trend in Research and Development*, vol. 5, no. 4, pp. 298–301, Aug. 2018.

[14] S. R. Gupta and C. P. Vora, "A Review Paper on Pressure Vessel Design and Analysis," *International Journal of Engineering Research*, vol. 3, no. 3, pp. 295–300, Mar. 2014.

[15] A. Akhmeteli and A. V. Gavrilin, "Vacuum Balloon–A 350-Year-Old Dream," *Eng*, vol. 2, no. 4, pp. 480–491, Dec. 2021, https://doi.org/10.3390/eng2040030.

[16] S. Salins, M. Mohan, and C. Stephen, "Finite Element Investigation on the Performance of Pressure Vessel Subjected to Structural Load," *Annales de Chimie - Science des Matériaux*, vol. 45, pp. 201–205, Jun. 2021, https://doi.org/10.18280/acsm.450302.

[17] S. Parkhe and K. Annamalai, "Design and Analysis of Pressure Vessel Subjected to Pressure-temperature Variation," *International Journal of Engineering*, vol. 31, no. 1, pp. 58–64, Jan. 2018, https://doi.org/10.5829/ije.2018.31.01a.09.

[18] K. Kazeem, A. F. Abdulfatai, and E. A. Edith, "Design and Analysis of a Vertical Pressure Vessel with Effect of Rotational Velocity on the Stresses and Deformation by using ANSYS," *International Journal of Analytical, Experimental and FEA*, pp. 110–120, Oct. 2019, https://doi.org/10.26706/ijaefea.2.6.20190702.

[19] N. Pervan, M. Trobradovic, A. J. Muminovic, H. Lulic, S. Metovic, and V. Hadziabdic, "Effects of Geometry Design Parameters on the Fatigue Failure of a Drive Axle Housing using FEA," *Engineering, Technology & Applied Science Research*, vol. 14, no. 1, pp. 12567–12573, Feb. 2024, https://doi.org/10.48084/etasr.6467.

[20] V. V. Wadkar, S. S. Malgave, D. D. Patil, H. S. Bhore, and P. P. Gavade, "Design and Analysis of Pressure Vessel Using ANSYS," *Journal of Mechanical Engineering and Technology (JMET)*, vol. 3, no. 2, pp. 01–13, Oct. 2015.

[21] Y.-J. Lu and C.-H. Wang, "A finite element-based analysis approach for computing the remaining strength of the pressure equipment with a local thin area defect," *Engineering Failure Analysis*, vol. 131, Jan. 2022, Art. no. 105883, https://doi.org/10.1016/j.engfailanal.2021.105883.

[22] E. Solangi, I. Memon, M. Memon, M. Raees, and M. Khan, "Failure Analysis of a Diesel Engine Crankshaft based on Finite Element Modeling," *Journal of Applied Engineering & Technology (JAET)*, vol. 7, pp. 1–9, Jun. 2023, https://doi.org/10.55447/jaet.07.01.82.