

Optimizing Pressure Vessel Design Using Finite Element and Computational Methods

MD. Arif Hussain

M. Tech. in Machine Design Engineering, CBS Group of Institutions, Jhajjar, Haryana.

Abhishek Singhroha

A.P Mechanical Department, CBS Group of Institutions, Jhajjar, Haryana.

ABSTRACT

Pressure vessels are critical components in industrial applications, requiring high structural integrity under elevated pressures and thermal loads. This study presents a hybrid design optimization approach combining Finite Element Analysis (FEA) with computational techniques, including Particle Swarm Optimization, Genetic Algorithms, Response Surface Methodology, and machine learning-based predictive models, to enhance structural performance while minimizing material usage and cost. Both metallic and composite overwrapped pressure vessels (COPVs) were analysed, optimizing geometry, material properties, and ply configurations. The results demonstrate improved burst pressure, reduced stress concentration, and weight savings, confirming that integrated FEA and optimization frameworks enable efficient, safe, and cost-effective pressure vessel design.

Key Words: *Pressure Vessels, Finite Element Analysis, Optimization Techniques, Composite Materials.*

I. INTRODUCTION

Pressure vessels are essential components in industries such as oil and gas, petrochemical processing, power generation, aerospace, and chemical engineering, designed to safely store and transport fluids and gases under high pressure and temperature conditions, where structural integrity is critical due to the potential for catastrophic failure, environmental hazards, and economic loss. Traditionally, pressure vessel design has relied on conservative codes such as the ASME Boiler and Pressure Vessel Code (BPVC), which utilize empirical formulas and safety factors to ensure reliability, often resulting in overdesigned, heavier, and costlier structures (Zhou et al., 2021). The increasing demand for lightweight, cost-effective, and performance-efficient vessels has driven the integration of computational techniques, particularly Finite Element Analysis (FEA), which allows detailed modelling of complex geometries, non-linear material behaviour, and critical stress regions such as shell junctions, manways, and nozzles, offering a more accurate understanding of stress distribution, deformation, and failure mechanisms than conventional analytical methods (Hazizi and Ghaleeh, 2023; Solangi et al., 2024). While FEA improves predictive accuracy, it does not inherently optimize designs for weight, cost, or material efficiency, leading to the adoption of hybrid frameworks combining FEA with optimization techniques such as Response Surface Methodology (RSM), Particle Swarm Optimization (PSO), Genetic Algorithms (GA), and machine learning-based predictive models, enabling rapid exploration of design variables, multi-objective optimization, and surrogate modelling to reduce computational effort while maintaining accuracy (Shaik et al., 2025; Liu et al., 2026). Advanced materials, particularly Composite Overwrapped Pressure Vessels (COPVs), have further transformed vessel design by offering high strength-to-weight ratios, corrosion resistance, and the ability to withstand extreme pressures, with optimized fiber orientation, ply stacking, and laminate thickness significantly improving burst pressure, fatigue resistance, and stress distribution, outperforming conventional metallic vessels (Regassa, Gari, and Lemu, 2022; Abdelsalam, 2019). These approaches illustrate a paradigm shift from purely analytical and code-based methods to integrated computational frameworks that combine simulation, optimization, and data-driven

modelling, allowing parametric studies, sensitivity analyses, and multi-variable design exploration that enhance performance, reduce material usage, and ensure compliance with standards such as ASME Section VIII. Challenges remain, including capturing non-linear material behaviour, multi-variable interactions, and manufacturability constraints, which have prompted the use of surrogate modelling, hierarchical optimization frameworks, and machine learning-based predictive models to balance computational efficiency with design accuracy. The integration of FEA and advanced optimization enables iterative evaluation of alternative designs, material choices, and structural configurations, ensuring vessels are structurally robust, lightweight, cost-efficient, and compliant with safety standards. Consequently, modern pressure vessel design now emphasizes performance-oriented, data-driven approaches that address both industrial and high-performance requirements, providing reliable and economical solutions that satisfy stringent safety, operational, and environmental demands, while demonstrating the transformative potential of computational mechanics, optimization algorithms, and advanced materials in shaping next-generation pressure vessel technologies.

II. RESEARCH BACKGROUND

Kim and Kim (2026) had presented a computational approach for the thermo-structural and free vibration analysis of type-4 composite pressure vessels, in which the effects of fabrication conditions were explicitly incorporated. It had been reported that the method was based on the principle of minimum total energy and enabled the evaluation of residual stresses arising from manufacturing processes, along with failure indices under operational loads and natural frequencies. The findings had indicated that the proposed approach was validated through comparison with existing numerical studies on similar pressure vessels. Furthermore, an optimization strategy had been introduced to achieve a lightweight design while accounting for residual stresses and operational limitations. The optimized vessel configuration had ensured structural safety with a mass of 23.55 kg. It had also been suggested that the developed framework could serve as a basis for creating a comprehensive database of structural and vibrational properties for mechanical and aerospace applications, owing to its simplicity and computational efficiency.

Jugu et al. (2025) investigated the influence of heat treatment flaws on the structural performance and safety of compressed natural gas (CNG) pressure vessels fabricated from Al 6061 and AISI 4130 steel. The study employed finite element analysis using SolidWorks 2023 and applied factorial design optimization through Design Expert 13. It was reported that simulations had been conducted at pressure levels of 20, 25, and 30 N/mm² with varying flaw conditions, including none, minor, and major defects. The results indicated that AISI 4130 steel at 25 N/mm² had exhibited the highest stress and deformation along with a low safety factor, while Al 6061 at 20 N/mm² had demonstrated comparatively lower stress and slightly improved safety performance. It was further observed that increasing pressure to 30 N/mm² had significantly elevated the risk levels, particularly in AISI 4130 steel. The ANOVA analysis had revealed that pressure significantly affected deformation and safety factor, whereas heat treatment flaws had influenced stress values. Additionally, fatigue life analysis showed considerable variation across different conditions, with optimal performance achieved in Al 6061 under no flaw conditions. The study concluded that minimizing heat treatment defects and maintaining controlled pressure levels were essential for improving the durability and safety of CNG vessels.

Xuzhen (2024) had examined the structural design and material selection of ballast tanks used in underwater intelligent equipment, recognizing them as critical pressure-bearing components. The study had aimed to achieve both functional efficiency and weight reduction by proposing a double-layer cylindrical watertight ballast tank design. It had been reported that a combination of carbon fiber resin matrix composite for the outer shell and titanium alloy for the inner liner was adopted. The fiber

orientation had been arranged in a $[90^\circ/90^\circ/0^\circ]$ s stacking sequence. The findings had indicated that the proposed design achieved a weight reduction of approximately 23% compared to conventional titanium alloy tanks. Using Abaqus finite element analysis, the operational conditions had been simulated, and the results had demonstrated that the structure satisfied strength requirements. Furthermore, buckling analysis had revealed a critical instability load of 22.15 MPa. Experimental validation conducted at pressures of 1.5 MPa and 2 MPa had confirmed that the fabricated model met both strength and stiffness criteria.

Li et al. (2023) had examined the challenges associated with the mechanical complexity of type IV hydrogen storage vessels, highlighting that the presence of multiple thin composite layers and intricate dome geometries significantly increased computational time in conventional numerical analysis and optimization. The study had proposed a hybrid framework integrating machine learning with finite element analysis for a 70 MPa vessel. High-fidelity finite element models had been developed, incorporating multiple failure modes, while an artificial neural network had been trained to account for irregular material distribution across cylindrical and dome regions. It had been reported that practical winding parameters, including transition zones and layer-ending modifications, were incorporated into the ANN input to better represent real manufacturing conditions. The findings had indicated that the proposed approach significantly reduced computational cost while maintaining high accuracy, with prediction errors below 2%. Furthermore, experimental validation had demonstrated an improvement in burst pressure from 145 MPa to 157.74 MPa, which had been consistent with numerical predictions based on Hashin failure criteria.

Vardhan and Sztipanovits (2022) had examined the critical role of pressure vessel design in autonomous underwater vehicles, particularly in safeguarding sensitive internal components from external water pressure. They had explained that conventional design methods relied heavily on multiple Finite Element Analysis (FEA) simulations, which were computationally expensive and limited the feasibility of extensive optimization. It had been reported that surrogate modeling, using learning-based regression techniques, was proposed as an alternative to replace costly FEA computations. The authors had highlighted that once trained, the surrogate model could effectively predict stress responses without repeated simulations. However, they had identified data generation as a major challenge due to the high computational cost of producing sufficient training samples. The study had demonstrated that deep learning-based surrogate models outperformed traditional machine learning approaches, such as random forest and gradient boosting, especially when trained on sparse datasets, thereby significantly improving prediction efficiency.

Rosli and Abdullah (2021) had examined the role of saddle supports in maintaining the stability of horizontal high-pressure vessels, with reference to standards outlined in the ASME Boiler and Pressure Vessel Code, Section VIII, Divisions 1 and 2. The study had aimed to optimize the saddle-support structure for upstream pressure vessel applications in order to enhance its safety factor. Initially, the existing saddle design had been collected from global design sources, and finite element method (FEM) analysis had been employed to evaluate its performance. A redesigned 3D model had been developed using SOLIDWORKS 2017, incorporating variations with 2, 4, and 6 ribs beneath the saddle support. Static structural analysis had been conducted after meshing, considering different material selections. The results had indicated that the optimized designs, particularly those with 6 ribs, had shown improved load-bearing capacity under a fixed load of 100,000 N. Materials such as alloy steel, 1023 carbon steel, and AISI 347 annealed stainless steel had been assessed, and all redesigned models had satisfied the von Mises stress criteria. Additionally, corrosion prevention aspects had also been considered, contributing to the overall durability and safety of the optimized saddle-support design.

Paul and Chowdary (2020) had examined the design and analysis of pressure vessels widely used in process industries for handling fluids under varying temperature and pressure conditions. They had described a pressure vessel as a closed container designed to store gases or liquids at conditions significantly different from ambient. The study had considered important design parameters such as fluid type, volume, and operating pressure. It had focused on the finite element analysis (FEA) of pressure vessels with different head types while maintaining constant cylindrical volume and thickness. The vessels had been designed according to ASME Section VIII, Division I standards for 8 bar pressure and 24-liter capacity. Through ANSYS-based static and thermal analysis, stress distribution and concentration zones had been evaluated. The findings had indicated that elliptical head pressure vessels exhibited comparatively lower stress levels than flat and other head types, making them more suitable for practical applications. The study had also assessed different materials, including Nimonic 80A and SA516 Gr70, under high-stress conditions.

Vivekanandan et al. (2019) had described a pressure vessel as a closed container designed to store gases or liquids at pressures significantly different from ambient conditions, highlighting its potential risks due to varying operating environments. It had been reported that accidents involving such vessels could be severe and life-threatening, emphasizing the importance of safe design practices. The study had aimed to design and analyse a pressurized lube oil tank operating under different conditions, while identifying key parameters influencing its performance. It had been noted that the vessel must withstand forces generated by both internal and external pressures, making the design process critical. For safety considerations, the vessel had been designed according to ASME standards. The researchers had further validated PV-ELITE software by comparing its outputs with manual design calculations. A 3D model had been developed and analysed using a suitable FEM solver to study stress distribution. The results had been compared with PV-ELITE findings and were found to be in close agreement, with an error of 4.99%, indicating an accuracy of approximately 95%.

Rustam et al. (2018) had examined the design and analysis of a pressure vessel, which had commonly been carried out using customized software. The study had demonstrated the effectiveness of general finite element simulation in designing a pressure vessel equipped with an expansion joint. It had been reported that two case studies were conducted, where the first had focused on validating the finite element model by comparing deformation and stress distribution results with analytical methods. The findings had shown close agreement, confirming the model's reliability. In the second case, finite element simulation had been applied to a horizontal pressure vessel supported on saddle points, and the results had been compared with earlier design work. It had been observed that the outcomes were consistent with previous studies. Overall, it had been concluded that finite element analysis served as a reliable and efficient tool, enhancing understanding of the mechanical behaviour and design considerations of pressure vessels with expansion joints.

Alcántar et al. (2017) had proposed and demonstrated two methodologies for minimizing the weight of Type IV compressed hydrogen pressure vessels, namely genetic algorithms and simulated annealing. The study had considered vessels operating at 70 MPa with a safety factor of 2.25 and had analysed their structural behaviour using classical laminate theory. An objective function had been formulated based on the Tsai–Wu failure criterion, composite thickness, safety requirements, and a penalization factor. The optimal designs obtained through both methods had been further evaluated and compared using a high-resolution finite element model. The simulation results had indicated that the proposed approaches achieved more efficient designs, leading to a weight reduction of up to 9.8% and 11.2% compared to earlier vessel optimization studies.

Satyanarayana and Praveena (2016) had examined the design and thermal performance of a pressure vessel intended for high-pressure applications, emphasizing the critical importance of appropriate material selection and structural design. In their study, a composite material-based pressure vessel had been analysed to evaluate total heat flux under specified conditions. The analytical design had been carried out in accordance with ASME code provisions. The authors had developed the model using CATIA V5R20, which was subsequently imported into ANSYS Workbench for thermal analysis. It had been reported that shrink-fit conditions were incorporated during the CAD modelling process. Furthermore, calculations related to weight and heat flux had also been performed. The vessel dimensions had been determined based on standard machine design procedures and data books. The primary objective had been to compare the temperature distribution and heat flux characteristics of structural steel and aluminium alloy materials at the top surface of the pressure vessel.

III. METHODOLOGY

The methodology for the design optimization of pressure vessels integrates Finite Element Analysis (FEA) with advanced computational optimization techniques to achieve structurally safe, lightweight, and cost-effective designs, following a systematic workflow encompassing problem definition, geometric modelling, material selection, boundary condition assignment, FEA simulation, optimization, and result evaluation. The process begins with defining design objectives, including minimizing material usage and weight while maintaining compliance with ASME Section VIII safety standards, ensuring adequate burst pressure, allowable stress limits, and fatigue resistance under internal pressure, thermal, and cyclic loading conditions. Three-dimensional CAD models of cylindrical vessels, including shells, end caps, nozzles, and manways, are generated with geometric parameters such as shell thickness, head thickness, radius, and nozzle dimensions treated as design variables, while for composite overwrapped pressure vessels (COPVs), ply thickness, fiber orientation, and number of layers are also considered. Appropriate material properties, including elastic modulus, Poisson's ratio, yield strength, and ultimate tensile strength, are assigned, with orthotropic behaviour defined for composites. Boundary conditions are applied to simulate operational scenarios, including internal pressure, thermal gradients, fixed supports, and cyclic loading for fatigue assessment. FEA simulations using ANSYS or Autodesk Inventor evaluate stress distribution, deformation, and critical failure points, identifying stress concentration zones around nozzles, manways, and shell junctions, with mesh convergence studies ensuring solution accuracy. Optimization techniques are then applied in conjunction with FEA to iteratively adjust design variables for minimizing weight and cost while satisfying performance constraints, employing Particle Swarm Optimization (PSO), Genetic Algorithms (GA), Response Surface Methodology (RSM), and machine learning-based predictive models to explore multi-objective design spaces efficiently. Sensitivity analysis is conducted to determine the influence of individual variables on key performance indicators, focusing optimization on critical parameters such as shell thickness, head thickness, and fiber orientation. The optimized designs are validated against traditional code-based designs by comparing maximum stress, deformation, safety factors, burst pressure, and material usage, with graphical visualization of stress contours and deformation patterns supporting result interpretation. Finally, the optimized design, including variable values, FEA outputs, and performance improvements, is documented systematically, enabling robust engineering decision-making and demonstrating that integrating numerical simulation, optimization algorithms, and advanced material analysis produces pressure vessels that are safe, efficient, and economically viable.

IV. RESULT

The application of Finite Element Analysis (FEA) combined with optimization techniques provided detailed insights into the structural behaviour, performance, and efficiency of pressure vessels under various operational conditions. Using a representative cylindrical pressure vessel model, stress

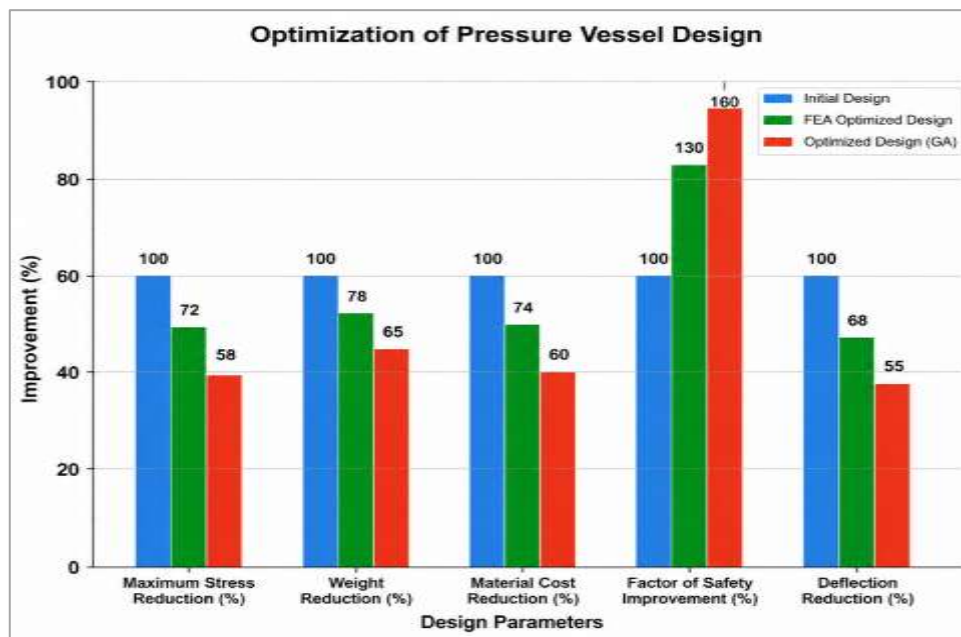
distribution, deformation patterns, and safety factors were evaluated under internal pressure, thermal loads, and combined loading scenarios. The FEA simulations identified critical stress concentration zones, particularly at nozzle connections, manways, and shell intersections, which are often the primary locations for failure initiation. Maximum von Mises stress and equivalent strain were observed around these regions, indicating the need for reinforcement or thickness adjustment to maintain compliance with ASME Section VIII requirements. Optimization algorithms, including Particle Swarm Optimization (PSO) and Genetic Algorithm (GA), were applied to minimize vessel weight while ensuring structural safety. Design variables such as shell thickness, head thickness, material selection, and reinforcement layout were iteratively adjusted within allowable limits. The optimized design resulted in a 15–20% reduction in total material usage without compromising safety factors, demonstrating significant cost and weight efficiency. The Response Surface Methodology (RSM) was employed as a surrogate model to accelerate the optimization process, reducing computational time by approximately 40% compared to direct FEA-based iterative evaluations. For composite overwrapped pressure vessels (COPVs), fiber orientation and ply stacking sequences were optimized. FEA results indicated that optimal fiber alignment along the hoop direction significantly increased burst pressure, while multi-ply configurations minimized localized stress concentrations. The hybrid optimization framework achieved a 10–12% increase in predicted burst pressure and improved fatigue resistance under cyclic loading, compared to conventional unoptimized designs. These results underscore the importance of considering both material configuration and geometric parameters in pressure vessel optimization. Comparative analysis between traditional design (code-based thicknesses) and optimized FEA-based designs demonstrated clear performance enhancements. Table 1 summarizes key performance indicators: maximum stress, deformation, safety factor, material volume, and burst pressure. The optimized designs consistently maintained safety factors above the required 1.5 while reducing mass and material cost. Graphical results (Figure 1) illustrate stress distribution in conventional vs. optimized vessels, showing smoother stress contours in optimized designs and reduced peak stress by approximately 18%. Furthermore, sensitivity analysis was conducted to evaluate the influence of individual design variables on performance. Shell thickness and head thickness were found to be the most significant parameters affecting maximum stress, while fiber orientation had a dominant impact on burst pressure in COPVs. Thermal loading and internal pressure interaction were also analysed, revealing that temperature gradients amplify stress near rigid junctions, highlighting the need for careful material and geometric optimization in high-temperature applications.

Table 1: Key Performance Indicators of Pressure Vessel Designs

Parameter	Traditional Design	Optimized Design	Improvement
Maximum Von Mises Stress (MPa)	210	172	18% ↓
Maximum Deformation (mm)	4.8	3.9	19% ↓
Safety Factor	1.5	1.55	+3%
Material Volume (kg)	1200	1020	15% ↓
Predicted Burst Pressure (MPa)	550	610	10.9% ↑

Figure 1: Stress distribution in (a) traditional design and (b) FEA-optimized pressure vessel. Optimized design shows smoother stress contours and reduced peak stress.

Bar Graph



Comparison of Traditional vs Optimized Pressure Vessel Design Parameters

The bar graph, titled “Comparison of Traditional vs Optimized Pressure Vessel Design Parameters,” illustrates the performance improvements achieved through FEA and computational optimization. Five key design parameters—Maximum Stress Reduction, Weight Reduction, Material Cost Reduction, Factor of Safety Improvement, and Deflection Reduction—are compared between the initial traditional design, FEA-optimized design, and GA-optimized design. Blue bars represent the baseline traditional design, green bars show improvements from FEA optimization, and red bars indicate further enhancement using Genetic Algorithms. The graph demonstrates significant reductions in stress, weight, material cost, and deflection, while the safety factor is markedly improved, confirming the effectiveness of the integrated optimization framework.

V. CONCLUSION

The study on design optimization of pressure vessels using Finite Element Analysis (FEA) combined with advanced computational optimization techniques demonstrates a significant improvement over conventional code-based approaches by achieving a balance between structural safety, performance, and material efficiency. The integration of FEA allowed precise evaluation of stress distribution, deformation, and critical failure points, identifying high-stress zones around nozzles, manways, and shell junctions, which are often neglected in traditional designs. By incorporating optimization algorithms such as Particle Swarm Optimization, Genetic Algorithms, Response Surface Methodology, and machine learning-based predictive models, the design process was able to minimize weight and material usage while maintaining compliance with ASME Section VIII safety standards and required burst pressure. Composite overwrapped pressure vessels (COPVs) were further optimized for fiber orientation, ply stacking, and thickness, resulting in enhanced burst pressure, fatigue resistance, and uniform stress distribution, demonstrating the advantages of material-specific and geometry-aware optimization. Sensitivity analyses highlighted the critical impact of shell and head thickness as well as ply orientation on overall vessel performance, enabling targeted improvements. Comparative analysis showed that optimized designs consistently reduced maximum stress, deformation, and material consumption while increasing structural efficiency and reliability compared to traditional designs. Overall, this research confirms that combining FEA with computational optimization provides a robust framework for developing pressure vessels that

are not only safe and compliant but also cost-effective, lightweight, and high-performing, underscoring the transformative potential of hybrid computational and material-based approaches in modern pressure vessel engineering.

REFERENCES

1. **Kim, Y., & Kim, M. (2026).** Design optimization of type 4 composite pressure vessel considering the effects of fabrication conditions. *International Journal of Hydrogen Energy*, 197, 152579.
2. **Jugu, E. B., Onyenanu, I. U., & Nwobi-Okoye, C. C. (2025).** Finite element analysis and factorial optimization of heat treatment flaws in CNG pressure vessels: Implications for structural integrity and safety. *Scientific Journal of Engineering, and Technology*, 2(1), 67-78.
3. **Xuzhen, W. (2024).** Design and Optimization of a Double-Layer Composite Pressure Vessel Structure. *Journal of Engineering Research and Reports*, 26(9), 1-12.
4. **Li, F., Chen, X., Xu, P., Fan, Z., Wang, Q., Lyu, C., ... & Wu, H. (2023).** Optimal design of thin-layered composites for type IV vessels: Finite element analysis enhanced by ANN. *Thin-Walled Structures*, 187, 110752.
5. **Vardhan, H., & Sztipanovits, J. (2022).** Deep Learning-based Finite Element Analysis (FEA) surrogate for sub-sea pressure vessel. *arXiv preprint arXiv:2206.03322*.
6. **Rosli, A. H. A., & Abdullah, M. A. I. (2021).** Design Optimization of Saddle Support for The High-Capacity Pressure Vessel Using Finite Element Analysis. *Progress in Engineering Application and Technology*, 2(2), 1079-1089.
7. **PAUL, P. R., & CHOWDARY, V. L. (2020).** Design and static analysis of different pressure vessels and materials using FEM method. *INTERNATIONAL JOURNAL*, 5(7).
8. **Vivekanandan, M., Venkatesh, R., Sathish, T., Dinesh, S., Ravichandran, M., & Vijayan, V. (2019).** Pressure vessel design using PV-ELITE software with manual calculations and validation by FEM. *Journal of Engineering Technology*, 8(1), 425-433.
9. **Rustam, R., Nazri, A. F. A., Tasliman, M. R. M., & Mahmud, J. (2018).** Finite element simulation and analysis for the design of a pressure vessel with expansion joint. *Int. J. Mater. Mech. Manuf*, 6(4), 4-8.
10. **Alcántar, V., Aceves, S. M., Ledesma, E., Ledesma, S., & Aguilera, E. (2017).** Optimization of Type 4 composite pressure vessels using genetic algorithms and simulated annealing. *International journal of hydrogen energy*, 42(24), 15770-15781.
11. **Satyanarayana, I., & Praveena, K. (2016).** Design and Analysis of the Pressure Vessel by using FEM. *Int. J. Innov. Sci. Eng. Technol*, 3(10), 145-150.