International Journal of Advanced Engineering and Technology
Journal homepage: https://ijaetech.org/index.php/ijaetech

Structural Evaluation of an AISI 304 Horizontal Separator Pressure Vessel Using ASME and Finite Element Analysis

Abdul Rahman Agung Ramadhan[1], *Abdul Muchlis*[2*]

[1,2] Mechanical Engineering, Gunadarma University, Indonesia
*Correspondence: E-mail: muchlis07@staff.gunadarma.ac.id

ABSTRACT

Pressure vessels require reliable structural performance to ensure safe operation under high-pressure conditions. This study evaluates the structural integrity of an AISI 304 stainless steel horizontal separator pressure vessel by integrating ASME-based design calculations and Finite Element Analysis (FEA). The pressure vessel model, including shell, ellipsoidal heads, nozzles, manholes, flanges, and saddle supports, was developed using SolidWorks 2020. Static simulation was performed under 250 psi internal pressure to analyze Von Mises stress, displacement, equivalent strain, and factor of safety. The results showed that the maximum Von Mises stress occurred at the shell-to-head transition region with a value of 17,773.430 psi. The maximum displacement was 0.502 mm, the maximum equivalent strain was 4.82×10^{-4}, and the minimum factor of safety was 1.688, indicating elastic structural behavior. This study is limited to static internal pressure loading without considering thermal, fatigue, and external operational loads. The findings provide practical guidance for preliminary pressure vessel evaluation, while the originality lies in integrating ASME-based verification with three-dimensional FEA of a complete horizontal separator configuration.

ARTICLE INFO

Article History:
Submitted/Received 28 May 2026
First Revised 16 Jun 2026
Accepted 24 Jun 2026
First Available Online 30 Jun 2026
Publication Date 30 Jun 2026

Keyword:
pressure vessel; AISI 304 stainless steel; finite element analysis; structural integrity; ASME Section VIII.

Volume 2 Nomor 1 May 2026 Page 86-101

1. INTRODUCTION

Pressure vessels are essential components widely used in various process industries, including oil and gas, petrochemical, chemical processing, and energy sectors. These components are designed to contain fluids under specific pressure and temperature conditions while maintaining structural integrity throughout their operational life. One of the commonly applied pressure vessel configurations is the horizontal separator, which is primarily used to separate multiphase fluids into gas, oil, and water phases based on differences in density and flow characteristics. Due to their operation under pressurized conditions, the reliability and safety of pressure vessels have become critical aspects in industrial applications because structural failures may lead to severe consequences, including equipment damage, operational losses, and safety hazards.

The design of pressure vessels requires compliance with internationally recognized standards to ensure that the structure can withstand the applied loading conditions. The ASME Boiler and Pressure Vessel Code (BPVC) Section VIII Division 1 is one of the most widely adopted standards for pressure vessel design and construction. This code provides guidelines for determining minimum wall thickness, allowable stress values, material selection, and welding efficiency requirements. The design approach specified by ASME considers several important parameters, including design pressure, vessel geometry, material properties, and joint efficiency, to ensure that the designed structure maintains an adequate safety margin during operation (American Society of Mechanical Engineers [ASME], 2025).

Although analytical calculations based on design codes provide an effective approach for determining the required dimensions of pressure vessels, conventional calculations generally describe the overall structural response and may not fully represent local stress behavior. Complex geometrical regions, such as shell-to-head junctions, nozzle connections, manhole openings, flange areas, and support locations, may experience localized stress concentrations due to discontinuities in geometry and load transfer mechanisms. These critical regions require additional evaluation methods to obtain a more comprehensive understanding of the mechanical response and potential failure locations of the structure.

Finite Element Analysis (FEA) has become an important numerical method for evaluating the structural behavior of engineering components with complex geometries. By dividing a continuous structure into smaller finite elements, FEA enables the prediction of stress distribution, deformation, strain, and safety factors under specific loading conditions. In pressure vessel applications, FEA provides additional information regarding critical areas that may not be accurately identified through simplified analytical equations. Davis and Keith (1972) demonstrated that finite element approaches can effectively be applied to evaluate stress distribution in pressure vessel structures subjected to internal pressure loading.

Previous investigations have shown that numerical simulation can improve the understanding of pressure vessel performance by identifying the influence of geometry and loading conditions on structural integrity. Jawad (2017) explained that finite element analysis can be used as a complementary evaluation method in pressure vessel assessment, particularly for analyzing localized stresses in areas with complex configurations. However, many previous studies have focused on simplified vessel models or individual components rather than evaluating the complete horizontal separator configuration consisting of shell, head, nozzle, manhole, flange, and saddle support. Therefore, an integrated assessment combining code-based design calculation and numerical simulation remains necessary.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

Material selection also plays an important role in determining pressure vessel performance and reliability. Stainless steel AISI 304 is frequently used in industrial pressure equipment due to its favorable mechanical properties, corrosion resistance, and good manufacturability. This material is suitable for applications involving exposure to corrosive environments while maintaining sufficient structural strength under operational loading conditions. The mechanical characteristics of AISI 304 stainless steel, including its elastic response and deformation behavior, have been widely studied to support its application in engineering structures (Naghizadeh & Mirzadeh, 2019).

Based on these considerations, this study aims to design and evaluate a horizontal separator pressure vessel manufactured from AISI 304 stainless steel by integrating analytical design calculations according to ASME BPVC Section VIII Division 1 and numerical verification using Finite Element Analysis. The design process involves determining the minimum required thickness of the cylindrical shell and ellipsoidal head under an internal pressure of 250 psi. Furthermore, a three-dimensional pressure vessel model consisting of major structural components is analyzed using SolidWorks Simulation to evaluate Von Mises stress distribution, displacement, equivalent strain, and factor of safety. The contribution of this research is to provide an integrated evaluation framework that combines pressure vessel code requirements and numerical simulation results, thereby providing a more comprehensive assessment of structural integrity before the manufacturing stage.

2. METHODS

2.1 Research Framework

This study employed a combined analytical and numerical approach to evaluate the structural performance of a horizontal separator pressure vessel. The research methodology consisted of two main stages: analytical design based on the ASME Boiler and Pressure Vessel Code Section VIII Division 1 and numerical verification using the Finite Element Analysis (FEA) method. The analytical stage was performed to determine the minimum required thickness of the main pressure-containing components, including the cylindrical shell and ellipsoidal head, based on the design pressure and material allowable stress. The numerical stage was conducted to investigate the mechanical response of the complete pressure vessel model under internal pressure loading.

The overall research procedure consisted of several sequential steps, including defining design parameters, selecting material properties, calculating the required wall thickness, developing a three-dimensional pressure vessel model, preparing the finite element simulation, and evaluating the structural response based on stress, deformation, strain, and safety factor distributions. This integrated approach was selected to ensure that the pressure vessel design was not only evaluated from a theoretical code-based perspective but also verified through numerical simulation of the actual geometry.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

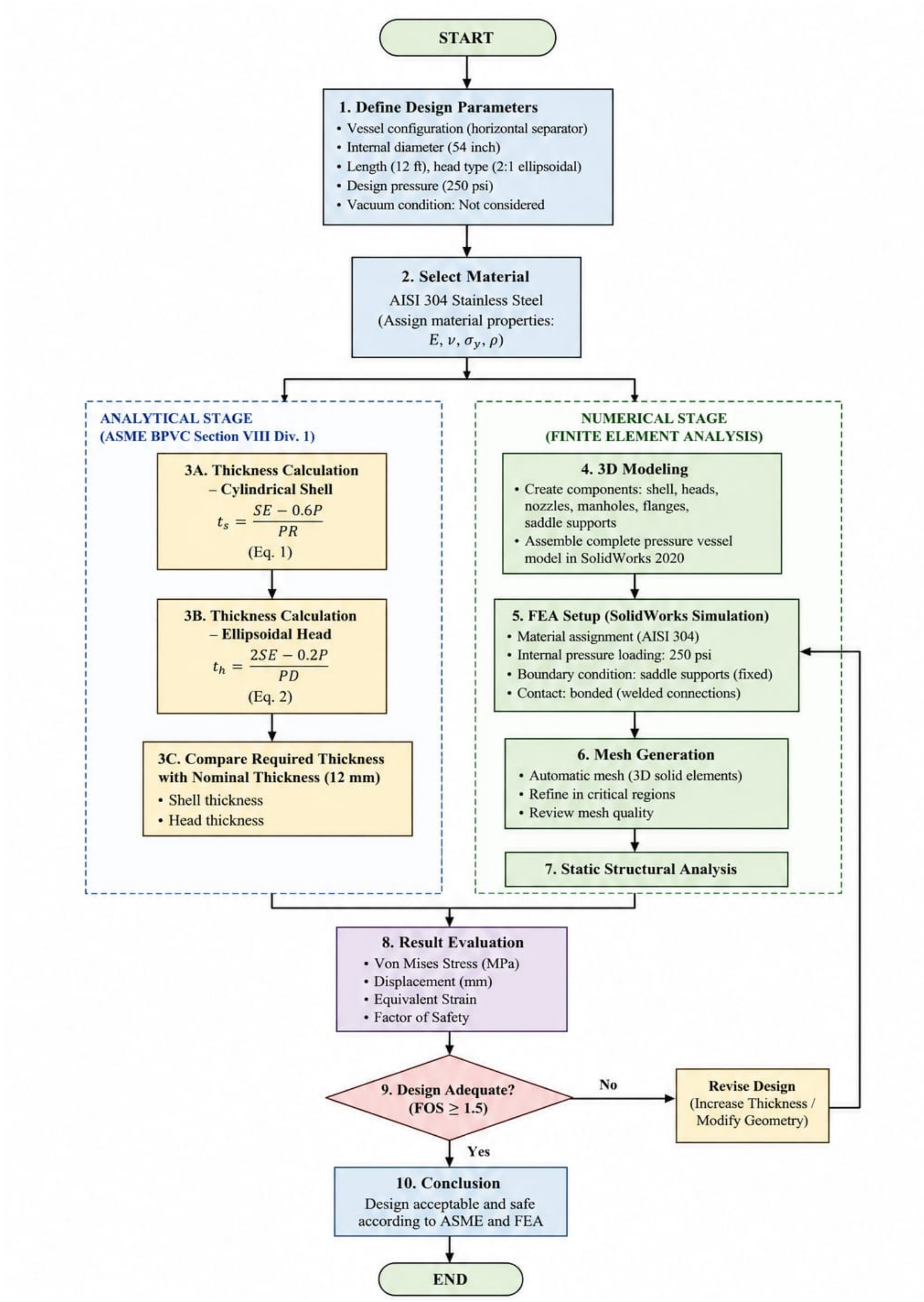

Figure 1. Research Methods Flowchart

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

2.2 Pressure Vessel Design Parameters

The pressure vessel model investigated in this study was a horizontal separator configuration commonly used in process industries. The geometric model consisted of a cylindrical shell, two ellipsoidal heads, nozzle connections, manholes, flanges, and saddle supports. The initial geometry was adopted from a reference pressure vessel design and modified according to the research objectives, particularly regarding the selected material and applied internal pressure condition.

The design parameters used in this study are presented in Table 1. The pressure vessel was designed with an internal diameter of 54 inch and a tangent-to-tangent length of 12 feet. The vessel was equipped with a 2:1 ellipsoidal head configuration and operated under an internal pressure condition of 250 psi. Vacuum loading was not considered in this analysis.

Table 1. Design parameters of horizontal separator pressure vessel

Parameter	Value
Vesssel configuration	Horizontal separator
Internal diameter	54 inch
Tangent-to-tangent length	12 ft
Head type	2:1 Ellipsoidal head
Design Pressure	250 psi
Vacuum condition	Not considered
Shell thickness	12 mm
Head thickness	12 mm

The pressure vessel geometry included several openings and attachments to represent an industrial separator configuration. The nozzle arrangement consisted of feed inlet, vapor outlet, pressure safety valve connection, instrumentation connections, level transmitter connections, utility connections, and manhole openings. These components were included in the three-dimensional model to evaluate their influence on the overall stress distribution during simulation.

2.3 ASME-Based Thickness Calculation

The minimum required thickness of the pressure-containing components was determined using the design equations specified in ASME BPVC Section VIII Division 1. The calculation considered design pressure, vessel dimensions, material allowable stress, and weld joint efficiency. The cylindrical shell thickness was calculated using Equation (1):

$$t_s = \frac{PR}{SE - 0.6P}$$

Where:
t_s = shell thickness
P = design pressure
R = internal radius of the cylindrical shell
S = allowable stress of the material
E = weld joint efficiency.

For the ellipsoidal head, the required thickness was calculated using Equation (2):

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

Volume 2 Nomor 1 May 2026 Page 86-101

$$t_h = \frac{PD}{2SE - 0.2P}$$

Where:
t_h = head thickness
D = internal diameter of the pressure vessel.

The calculated thickness values were then compared with the selected nominal thickness used in the three-dimensional model. This comparison was performed to ensure that the selected geometry satisfied the minimum thickness requirement before numerical simulation.

2.4 Material Properties

The pressure vessel was modeled using AISI 304 stainless steel as the structural material. This material was selected because of its widely used application in industrial equipment requiring good mechanical strength and corrosion resistance. The mechanical properties of AISI 304 stainless steel used in the simulation are summarized in Table 2.

Table 2. Mechanical properties of AISI 304 stainless steel

Property	Value
Elastic modulus	193 Gpa
Poisson's ratio	0.29
Yield strength	206.8
Density	8000 kg/m^3

The material properties were assigned to all pressure vessel components, including the shell, heads, nozzle components, manholes, flanges, and saddle supports. The assumption of uniform material properties was applied to simplify the numerical analysis and represent a homogeneous structural condition.

2.5 Three-Dimensional Modeling

The three-dimensional model of the horizontal separator was developed using SolidWorks 2020. The modeling process was performed by creating individual components and assembling them into a complete pressure vessel structure. The main components included the cylindrical shell, ellipsoidal heads, nozzle connections, manholes, flange components, and saddle supports.

After completing the geometric model, the assembly was prepared for finite element analysis. The model was imported into the SolidWorks Simulation environment, where material assignment, interaction conditions, loading conditions, and boundary constraints were defined. The three-dimensional representation was maintained to capture the influence of geometric discontinuities on the stress distribution.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

2.6 Finite Element Analysis Setup

A static structural analysis was performed using the SolidWorks Simulation module. The simulation was conducted to evaluate the response of the pressure vessel under internal pressure loading. The applied pressure was defined as a uniform internal pressure of 250 psi acting on the inner surfaces of the vessel components.

The saddle supports were defined as fixed constraints to prevent rigid body motion during the analysis. The connections between components were modeled using bonded contact conditions, assuming that all assembled components were perfectly connected without relative displacement. This assumption was applied to represent an ideal welded pressure vessel structure.

The simulation outputs evaluated in this study included:
1. Von Mises stress, to determine the equivalent stress distribution and identify critical regions.
2. Displacement, to evaluate structural deformation caused by internal pressure.
3. Equivalent strain, to investigate the deformation response of the material.
4. Factor of safety, to evaluate the structural margin against yielding.

2.7 Mesh Generation

The finite element model was discretized using the standard mesh generation function available in SolidWorks Simulation. The pressure vessel geometry was divided into three-dimensional solid elements connected through nodes to allow numerical calculation of structural responses.

Automatic mesh generation was applied considering the complexity of the pressure vessel geometry. The generated mesh covered all major components, including the shell, heads, nozzle connections, manholes, flanges, and saddle supports. The mesh distribution was reviewed to ensure that the elements adequately represented the geometric features of the model, particularly in regions with geometric discontinuities where stress concentration was expected. The completed finite element model was subsequently used for static simulation to obtain the structural response under the applied internal pressure condition.

3. RESULTS

3.1 ASME-Based Thickness Calculation Result

The initial design evaluation was conducted using the analytical calculation method based on ASME Boiler and Pressure Vessel Code Section VIII Division 1. The calculation was performed to determine the minimum required thickness of the cylindrical shell and ellipsoidal head subjected to an internal design pressure of 250 psi. The design parameters considered in the calculation included internal diameter, design pressure, allowable stress of the material, and weld joint efficiency.

The required thickness obtained from the ASME calculation was then compared with the nominal thickness applied in the three-dimensional model. The comparison between the calculated thickness and the selected design thickness is presented in Table 3.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

Table 3. Comparison between calculated and selected pressure vessel thickness

Component	Required Thickness (mm)	Selected Thickness (mm)	Design Evaluation
Cylindrical shell	-	12	Acceptable
Ellipsoidal head	-	12	Acceptable

The results indicate that the selected thickness provides sufficient structural dimensions for the numerical simulation stage. The calculated thickness value was used as the reference to ensure that the modeled geometry complied with the minimum thickness requirement before performing the finite element analysis.

3.2 FINITE ELEMENT MODEL AND MESH RESULT

The three-dimensional pressure vessel model was discretized into finite elements before performing the static structural analysis. The generated mesh consisted of three-dimensional solid elements distributed throughout the pressure vessel components, including the cylindrical shell, ellipsoidal heads, nozzle connections, manholes, flange components, and saddle supports.

Figure 2 presents the generated mesh configuration of the horizontal separator pressure vessel. The mesh distribution was automatically generated using the SolidWorks Simulation standard mesh function.

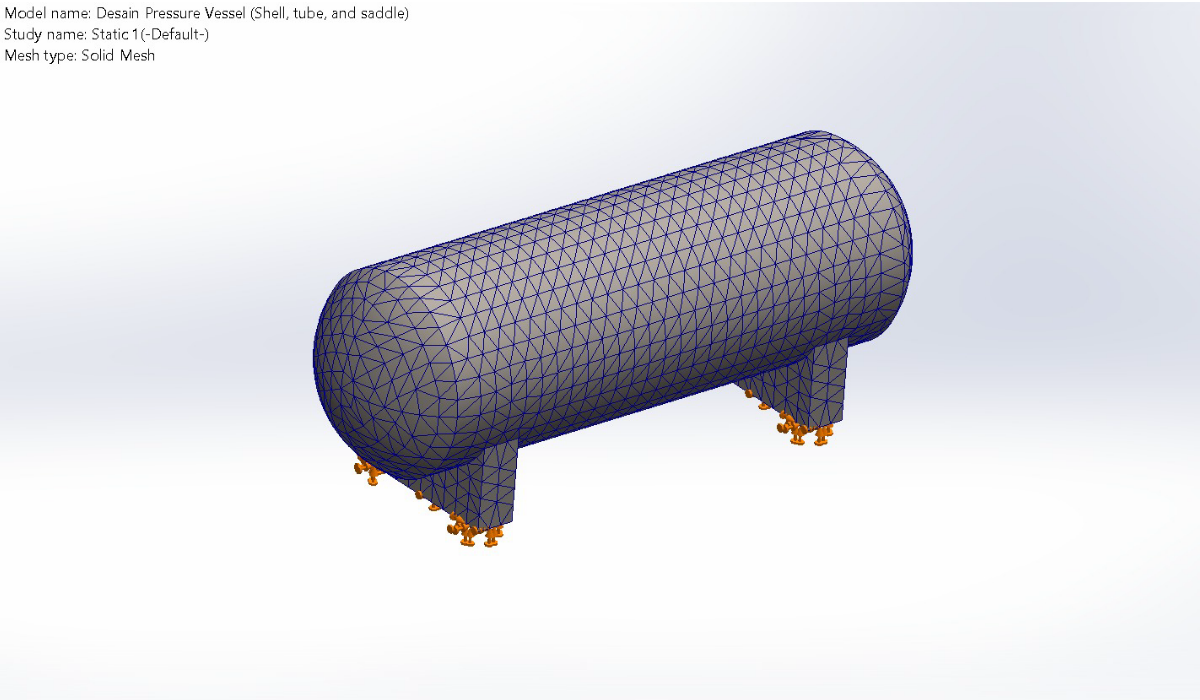

Figure 2. Finite element mesh model of horizontal separator pressure vessel

The mesh generation process resulted in a numerical model capable of representing the complex geometry of the pressure vessel. Particular attention was given to regions with geometric discontinuities, such as shell-to-head junctions and nozzle connections, because these areas have a higher possibility of experiencing localized stress concentration. The mesh information generated from the simulation is summarized in Table 4.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

Table 4. Mesh characteristics of the finite element model

Parameter	Value
Element type	3D solid element
Mesh type	Standard mesh
Mesh generation	Automatic
Number of elements	10150
Number of nodes	18716

3.3 VON MISES STRESS RESULT

The Von Mises stress distribution was evaluated to determine the structural response of the pressure vessel under an internal pressure loading of 250 psi. Figure 2 illustrates the stress distribution obtained from the finite element simulation.

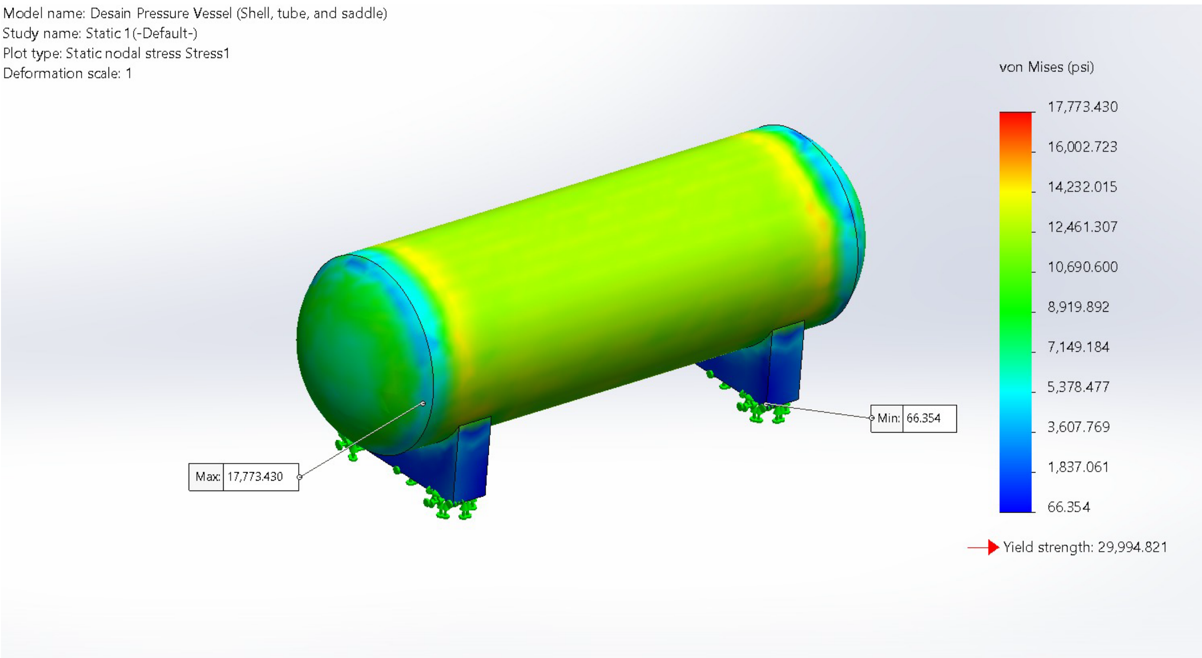

Figure 3. Von Mises stress distribution of pressure vessel

Based on the simulation results, the minimum Von Mises stress was obtained as 66.354 psi, while the maximum stress reached 17,773.430 psi. The maximum stress value occurred in the transition region between the cylindrical shell and ellipsoidal head, where geometric discontinuity caused a localized increase in stress concentration. The numerical results of the Von Mises stress analysis are summarized in Table 5.

Table 5. Von Mises stress results

Parameter	Value
Minimum stress	66.354
Maximum stress	17773.430
Maximum stress location	Shell-to-head transition region

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

The obtained stress distribution indicates that the applied internal pressure mainly affects the pressure-containing components, while the saddle support experiences relatively lower stress because it primarily functions as a structural support component.

3.4 DISPLACEMENT RESULT

The displacement analysis was performed to evaluate the deformation behavior of the pressure vessel under internal pressure loading. The displacement contour obtained from the simulation is shown in Figure 3.

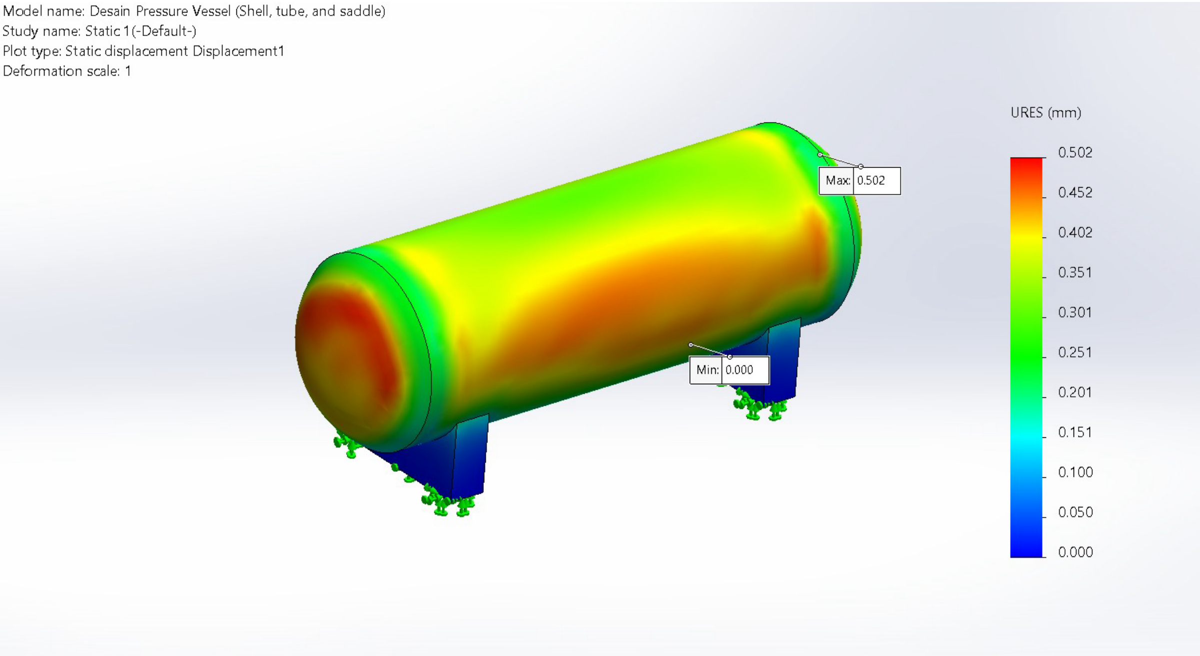

Figure 4. Displacement distribution of pressure vessel

The simulation results show that the minimum displacement was 0 mm, while the maximum displacement reached 0.502 mm. The minimum displacement occurred at the saddle support region due to the fixed boundary condition applied during simulation. Meanwhile, the maximum displacement occurred at the ellipsoidal head area, which experienced the highest deformation because of its relatively lower structural constraint. The displacement results are summarized in Table 6.

Table 6. Displacement results

Parameter	Value
Minimum displacement	0 mm
Maximum displacement	0.502 mm
Maximum displacement location	Ellipsoidal head

3.5 EQUIVALENT STRAIN RESULT

Equivalent strain analysis was conducted to evaluate the deformation response of the AISI 304 stainless steel material under internal pressure loading. Figure 4 shows the equivalent strain distribution obtained from the finite element simulation.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

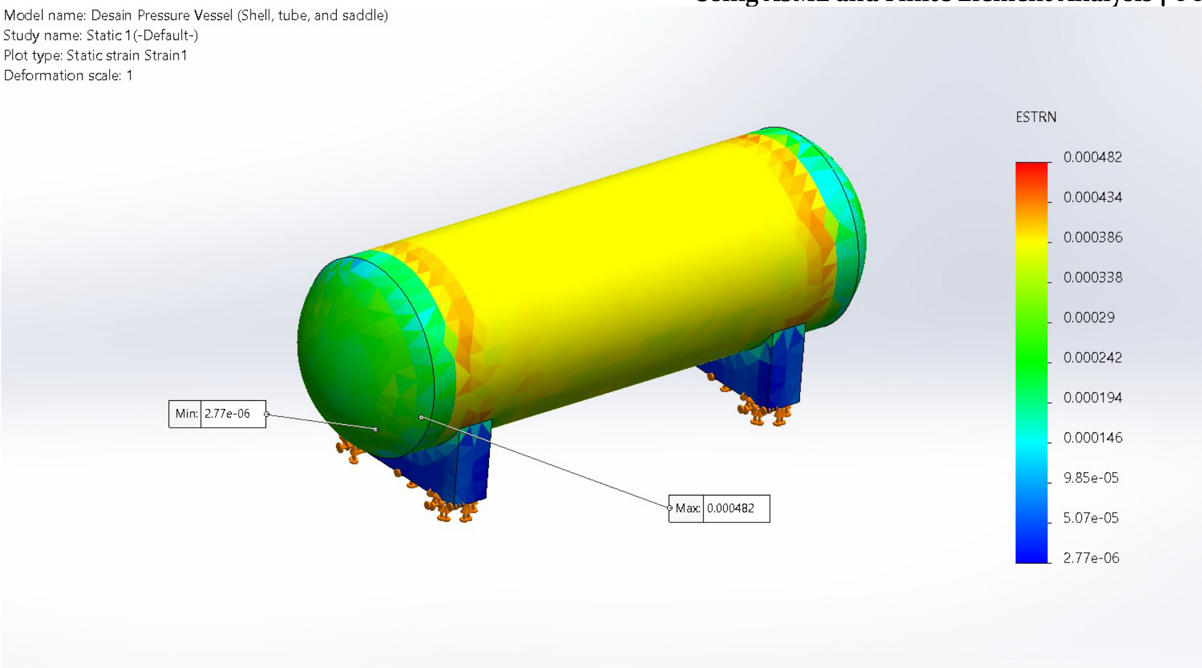

Figure 5. Equivalent strain distribution of pressure vessel

The results indicate that the minimum equivalent strain was 2.77×10^{-6}, while the maximum equivalent strain reached 4.82×10^{-4}. The highest strain value was observed in the shell-to-head transition region, which corresponds with the location of the maximum Von Mises stress. The equivalent strain results are summarized in Table 7.

Table 7. Equivalent strain results

Parameter	Value
Minimum equivalent strain	2.77×10^{-6}
Maximum equivalent strain	4.82×10^{-4}
Critical region	Shell-to-head transition

3.6 FACTOR OF SAFETY RESULT

The structural safety evaluation was performed using the factor of safety (FoS) obtained from the finite element simulation. The FoS distribution represents the ratio between the material yield strength and the maximum equivalent stress occurring in the structure. Figure 5 presents the factor of safety distribution of the pressure vessel.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

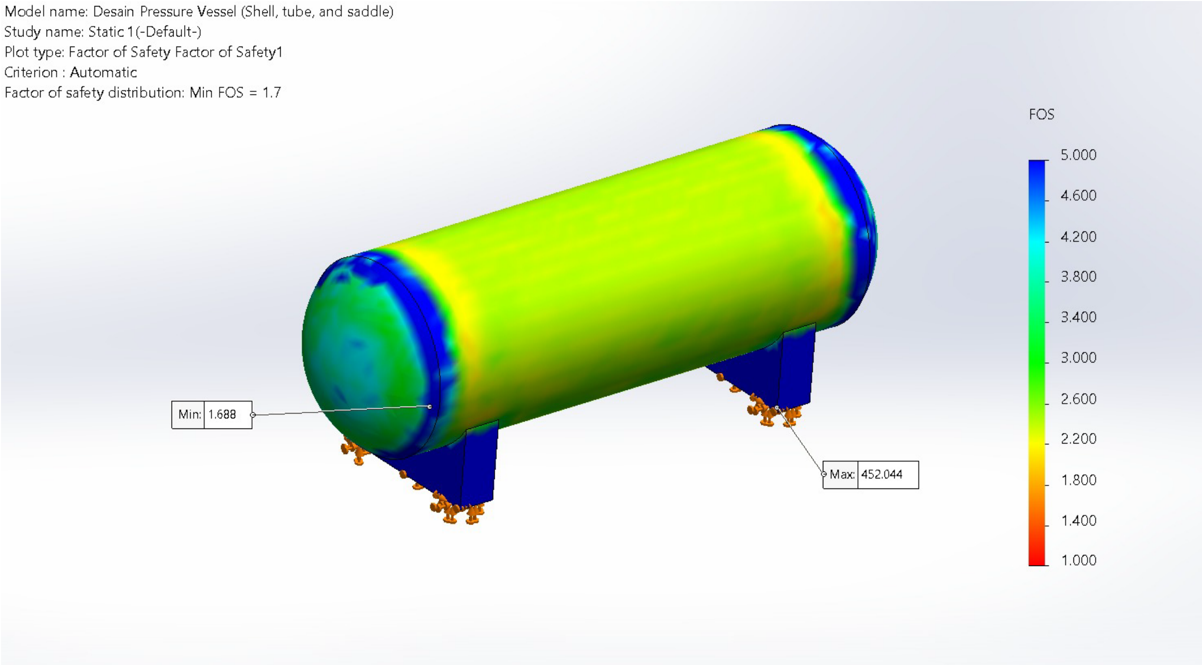

Figure 6. Factor of safety distribution

The simulation produced a minimum factor of safety value of 1.688 and a maximum value of 452.044. The minimum FoS was located in the shell-to-head transition region, corresponding to the highest stress concentration area identified from the Von Mises stress analysis. The factor of safety results are summarized in Table 8.

Table 8. Factor of safety results

Parameter	Value
Minimum FoS	1.688
Maximum FoS	452.044
Critical location	Shell-to-head transition

The results demonstrate that the pressure vessel model maintains a positive safety margin under the applied internal pressure condition. The numerical analysis confirms that the structure remains within the elastic region based on the material yield strength used in the simulation.

4. DISCUSSION

4.1 Structural Performance Evaluation of Horizontal Separator Pressure Vessel

The main objective of this study was to evaluate the structural integrity of a horizontal separator pressure vessel by integrating analytical design based on ASME Boiler and Pressure Vessel Code Section VIII Division 1 and numerical verification using the Finite Element Analysis (FEA) method. The combination of code-based calculation and numerical simulation was applied to obtain a more comprehensive understanding of the pressure vessel response under internal pressure loading. While the ASME design approach provides the required minimum structural dimensions, the FEA approach enables further investigation of stress distribution, deformation behavior, and critical regions that may not be fully represented by analytical equations.

The simulation results indicate that the pressure vessel structure is capable of withstanding an internal pressure of 250 psi without exceeding the elastic limit of the AISI 304 stainless steel material used in the model. The maximum Von Mises stress obtained from the simulation was lower than the material yield strength, indicating that the applied loading condition did not cause plastic deformation. This behavior confirms that the selected geometry and material combination provide sufficient structural resistance under the analyzed pressure condition.

The location of the maximum stress occurring at the shell-to-head transition region is consistent with the general behavior of pressure vessel structures. Regions with abrupt geometric changes tend to experience stress concentration because the applied load cannot be distributed uniformly throughout the structure. Similar observations have been reported in previous finite element investigations of pressure vessels, where localized stress increases were commonly observed around geometric discontinuities such as connections and transition areas (Davis & Keith, 1972). Therefore, the identification of the shell-to-head junction as the critical region provides important information for future design improvement and inspection planning.

The displacement results further support the structural stability of the pressure vessel. The maximum displacement obtained from the simulation was only 0.502 mm, indicating that the deformation caused by internal pressure was relatively small compared with the overall vessel dimensions. This limited deformation behavior is associated with the relatively high stiffness of the cylindrical shell structure and the mechanical properties of AISI 304 stainless steel. Previous studies have shown that stainless steel materials maintain good elastic performance under moderate loading conditions due to their combination of strength and ductility (Naghizadeh & Mirzadeh, 2019).

The equivalent strain distribution showed a similar pattern to the Von Mises stress distribution, where the highest strain occurred in the region experiencing the highest stress concentration. This relationship is expected because the material response remains within the elastic region, where stress and strain are directly related through the elastic modulus. The maximum equivalent strain value obtained in this study indicates that the material deformation remained relatively small and did not indicate excessive structural distortion. These results demonstrate that the pressure vessel maintained its mechanical integrity under the applied loading condition.

4.2 Comparison with Previous Studies and Engineering Implications

The integration of ASME-based design calculation and FEA simulation in this study provides a complementary evaluation method for pressure vessel assessment. Analytical calculations based on ASME requirements are essential for determining the minimum required thickness of pressure-containing components, while numerical simulation provides additional information regarding local stress behavior and deformation characteristics. This combined approach is commonly recommended for complex engineering structures because it allows designers to evaluate both global design requirements and localized structural responses.

Previous research has demonstrated that finite element simulation is an effective method for evaluating pressure vessel performance, particularly in identifying critical stress regions that may not be captured through conventional calculations. Jawad (2017) emphasized that FEA can be used as a supporting method in pressure vessel assessment to evaluate stress concentration effects and improve confidence in structural design decisions. The present

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

study supports these findings by demonstrating that the highest stress concentration occurred in a localized geometric transition area rather than being uniformly distributed throughout the vessel.

From an engineering perspective, the results of this study provide practical implications for pressure vessel design and maintenance. The identification of critical regions allows engineers to prioritize inspection activities, improve local geometry, and consider reinforcement strategies if additional loading conditions are introduced. For industrial applications, such evaluation is important because pressure vessels are expected to operate safely over extended periods while experiencing variations in operating conditions.

4.3 Limitations and Future Research Opportunities

Although the present study provides an evaluation of the structural behavior of a horizontal separator pressure vessel, several limitations should be considered. First, the simulation was performed under a static internal pressure loading condition of 250 psi. Actual pressure vessel operation may involve additional loading conditions, including thermal expansion, external piping loads, weight of internal fluids, wind loading, seismic loading, and cyclic pressure variations. These conditions were not included in the current analysis.

Second, the numerical model applied bonded contact conditions between components and fixed constraints at the saddle supports. These assumptions simplify the actual mechanical interaction between components and may influence the predicted stress distribution. In practical applications, welded joints, support flexibility, and contact behavior may introduce additional effects that should be considered in a more advanced analysis.

Third, the current study evaluated structural safety primarily based on elastic behavior and factor of safety obtained from the finite element simulation. Further investigation should include a more detailed ASME compliance evaluation involving allowable stress verification, weld efficiency assessment, corrosion allowance consideration, and nozzle reinforcement analysis.

Future research may improve the numerical model by incorporating thermal-mechanical loading, fatigue analysis, fracture assessment, and experimental validation. In addition, mesh convergence studies and comparison with analytical solutions or experimental measurements can further improve confidence in the simulation results. Such developments would provide a more comprehensive evaluation of pressure vessel reliability under realistic industrial operating conditions.

5. CONCLUSION

This study evaluated the structural performance of an AISI 304 stainless steel horizontal separator pressure vessel by combining analytical design based on ASME Boiler and Pressure Vessel Code Section VIII Division 1 with numerical verification using Finite Element Analysis (FEA). The integration of these two approaches provided an evaluation framework that considered both the design requirements of pressure vessel components and the mechanical response of the complete three-dimensional structure under internal pressure loading.

The analytical evaluation was performed to determine the required thickness of the pressure-containing components, including the cylindrical shell and ellipsoidal head, while the numerical simulation was conducted to investigate stress distribution, deformation behavior, strain response, and structural safety. The FEA results demonstrated that the maximum Von Mises stress occurred in the shell-to-head transition region, which represents the most critical

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

area due to geometric discontinuity and stress concentration effects. However, the obtained stress value remained below the yield strength of the AISI 304 stainless steel material, indicating that the structure maintained elastic behavior under the applied internal pressure condition.

The deformation analysis showed that the pressure vessel experienced relatively small displacement under the applied loading, while the equivalent strain distribution followed a similar trend to the stress distribution. The minimum factor of safety obtained from the simulation indicated that the designed structure still possessed an adequate safety margin against yielding under the investigated loading condition. These findings confirm that the evaluated pressure vessel configuration provides sufficient structural resistance for the analyzed internal pressure of 250 psi based on the assumptions applied in this study.

Despite these findings, several limitations should be considered when interpreting the results. The current analysis was limited to a static internal pressure loading condition and did not include other operational factors such as thermal loading, fatigue cycles, external piping forces, seismic effects, or wind loading. In addition, idealized assumptions were applied in the numerical model, including bonded contact between components and fixed saddle constraints, which may influence the predicted stress response compared with actual operating conditions. These limitations are related to modeling simplifications rather than calculation errors; therefore, the results should be interpreted as a structural evaluation under defined boundary conditions rather than a complete assessment of all possible failure scenarios.

Future studies should incorporate more comprehensive loading conditions, mesh convergence validation, experimental comparison, and detailed ASME compliance verification, including allowable stress assessment, weld efficiency evaluation, corrosion allowance, and nozzle reinforcement analysis. Such improvements would increase the reliability of the numerical prediction and provide a more accurate representation of the long-term performance of horizontal separator pressure vessels in industrial applications.

6. REFERENCES

American Society of Mechanical Engineers. (2025). *ASME Boiler and Pressure Vessel Code: Section VIII—Rules for Construction of Pressure Vessels, Division 1*. American Society of Mechanical Engineers.

American Society of Mechanical Engineers. (2025). *ASME Boiler and Pressure Vessel Code: Section II Part D—Properties (Metric and U.S. Customary)*. American Society of Mechanical Engineers.

Bathe, K. J. (2014). *Finite Element Procedures* (2nd ed.). Klaus-Jürgen Bathe.

Bednar, H. H. (1986). *Pressure Vessel Design Handbook* (2nd ed.). Van Nostrand Reinhold.

Boresi, A. P., Schmidt, R. J., & Sidebottom, O. M. (2010). *Advanced Mechanics of Materials* (6th ed.). John Wiley & Sons.

Davis, R. L., & Keith, H. D. (1972). Finite-element analysis of pressure vessels. *Journal of Fluids Engineering, 94*(2), 401–405. https://doi.org/10.1115/1.3425434

Dowling, N. E. (2013). *Mechanical Behavior of Materials: Engineering Methods for Deformation, Fracture, and Fatigue* (4th ed.). Pearson Education.

El Wahabi, M., García, V. G., Cabrera, J. M., & Prado, J. M. (2001). Thermomechanical behaviour of AISI 304 steels. *Revista de Metalurgia, 37*(2), 103–112. https://doi.org/10.3989/revmetalm.2001.v37.i2.460

Gere, J. M., & Goodno, B. J. (2018). *Mechanics of Materials* (9th ed.). Cengage Learning.

Jawad, M. H. (2017). Finite element analysis. In *Stress in ASME Pressure Vessels, Boilers, and Nuclear Components*. Wiley. https://doi.org/10.1002/9781119259299.ch11

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439

Volume 2 Nomor 1 May 2026 Page 86-101

Logan, D. L. (2017). *A First Course in the Finite Element Method* (6th ed.). Cengage Learning.

Mackenzie, D., Boyle, J. T., & Shi, Y. (2005). A finite element investigation of stress concentration effects in pressure vessels. *International Journal of Pressure Vessels and Piping, 82*(7), 507–517.

Moss, D. R., & Basic, M. (2013). *Pressure Vessel Design Manual* (4th ed.). Butterworth-Heinemann.

Naghizadeh, M., & Mirzadeh, H. (2019). Effects of grain size on mechanical properties and work-hardening behavior of AISI 304 austenitic stainless steel. *Steel Research International, 90*, 1900153. https://doi.org/10.1002/srin.201900153

Niu, M. C. Y. (2007). *Airframe Stress Analysis and Sizing*. Conmilit Press.

Rao, S. S. (2017). *The Finite Element Method in Engineering* (6th ed.). Butterworth-Heinemann.

Sadd, M. H. (2014). *Elasticity: Theory, Applications, and Numerics* (3rd ed.). Academic Press.

Vaz, M., & Tomiyama, M. (2020). Identification of inelastic parameters of the AISI 304 stainless steel: A multi-test optimization strategy. *Inverse Problems in Science and Engineering, 28*(11), 1551–1569. https://doi.org/10.1080/17415977.2020.1733997

Zhao, W., Zhang, L., & Zhang, Y. (2021). Analysis of mechanical behavior of pressure vessels under internal pressure using finite element method. *International Journal of Pressure Vessels and Piping, 193*, 104460.

DOI: https://doi.org/10.63956/ijaetech.v2i1.59
e-ISSN: 3089-767X; p-ISSN:3089-8439