STRESS ANALYSIS IN VESSELS USING FINITE ELEMENT METHOD

Isro Hadi
National Institute of Science and Technology Jakarta

isrohadi@gmail.com

ABSTRACT

The structural integrity and reliability of pressure vessels play an important role to ensure safety and optimal performance. The structural strength is calculated from the maximum stresses that occur in a pressure vessel based on the operating loads it receives. Technological advances have provided an option in performing structural strength analysis with the finite element method, a numerical computation that allows engineers to accurately predict the behavior of complex structures subjected to various load conditions. This research aims to obtain design results and design strength calculations on pressure vessels using ASME section VIII standards. Horizontal type pressure vessel with SS 400 steel material and yield strength of 235 MPa is analyzed using ANSYS Student Edition. The analysis results show that the maximum stress occurs at the base of the pressure vessel in direct contact with the fluid. The analyzed pressure vessel has met the necessary criteria and is suitable for production at static loading up to 2.5 Mpa where the specimen deforms about 0.17613 mm with a strain of about 0.0011559, and a stress of about 229.38 MPa. However, at 3 MPa loading, the stress has exceeded the material's allowable stress of 235 MPa. This study has a safety factor (SF) of 1 with a yield strength (YS) of 235 MPa. This means that the pressure vessel is designed to withstand loads that produce a maximum stress equal to YS without undergoing permanent plastic deformation.

Keywords: Pressure vessel, stress analysis, finite element method

1. INTRODUCTION

Voltage or Mechanical stress on an object is a physical quantity that shows the internal force between particles in an object due to the external force acting on the object. As a result of the force that causes external stress, the object can change shape or be deformed. The external force that works can be in the form of contact force, friction, pulling, pressing, and so on. In its definition, if the external force has a direction perpendicular to the surface area of the object, then such stress is called normal stress. Furthermore, if the external force that works has a direction parallel to the surface area of the object, it is called shear stress . The magnitude stress in a way general defined as force per unit wide (Santoso, 2017) .

In pressure vessel design, attention to stress analysis on the vessel wall is very important to ensure safety and optimal performance. Stresses that occur on the vessel wall can be caused by various factors, both external such as wind and earthquake loads, and internal such as working pressure and the weight of the vessel itself. Through careful stress analysis, we can determine the magnitude and position of the maximum stress that may occur. This is very crucial to determine the appropriate dimensions and types of materials in order to prevent material failure that can cause damage to the pressure vessel. Some factors that need to be considered in this analysis involve the type of fluid to be accommodated, working pressure, vessel dimensions, material type, and external factors such as wind and earthquake loads. By considering all these aspects, stress analysis on pressure vessels can be carried out accurately, avoiding potential material failure that can impact serious about safety humans and the environment (Manullang, E. et al., 2016) .

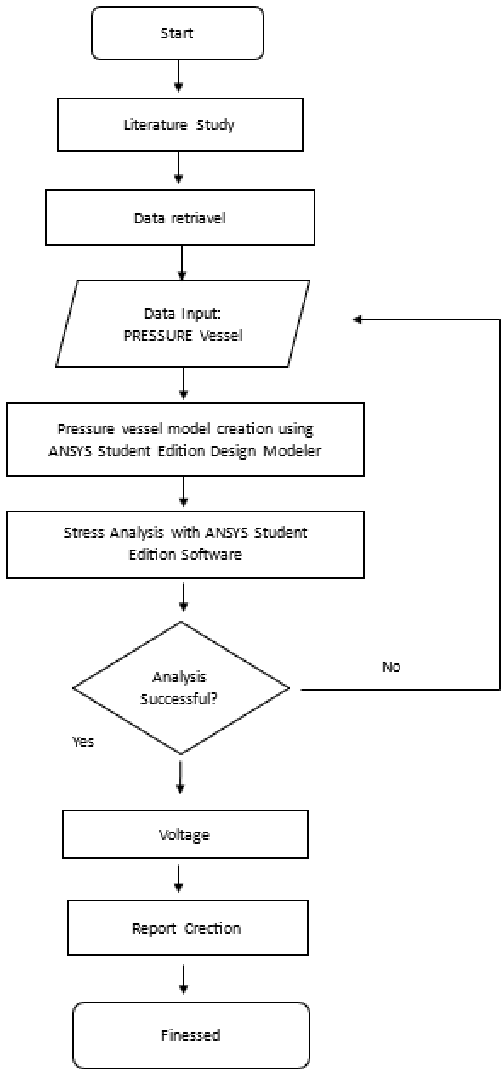
Pressure vessels have complex geometric structures consisting of various geometric discontinuities and are generally used to work under conditions high load like style external , thermal loads , internal pressures , etc. (Kumar, V. et al., 2014)). Load distribution in pressure vessels can be analyzed by the finite element method (Yang et al ., 1994). Product design and planning uses rules and codes according to international standards. In general, the standard codes used for design are ASME boiler pressure vessel section VII, European Committee for Standardization and British Standards Institution (Kumar, V. et al., 2014) Research about structural and reliability of the vessel press For avoid vessel malfunction press has Lots found (Purnomo, J. & Satrijo, 2012) ; (Effendi, 2014) ; (Manullang, E. et al., 2016) ; (Huda, K. & Permadi, 2017) ; (Nicholasta, KL et al., 2023) . Research using design software such as ANSYS and the finite element method (FEM) has helped visualize the distribution of stress, strain, and deformation on the walls of pressure vessels. The maximum stress occurs at the joint area, and the maximum stress value must be lower than the allowable stress of the vessel material.

Progress technology has provides an option in conducting structural strength analysis with the Finite Element Method (FEM), a numerical computation that allows engineers to accurately predict the behavior of complex structures subjected to various load conditions. This study aims to obtain the results of the design and calculation of design strength on pressure vessels using the ASME section VIII standard that is safe using software ANSYS Student Edition . The advantage of the ANSYS program compared to similar programs that use the FEM basis is the ease of program input. This program requires input data in the form of element types, loads, and geometry, after which the matrix equation is composed and the program can be run to perform calculations. The results of ANSYS calculations can be displayed numerically or visually, such as temperature and stress distributions, facilitating analysis. This advantage makes ANSYS suitable for analysis structure component technology temperature high , like in guide use ANSYS (Nakasone et al., 2018).

2. RESEARCH METHODS

2.1. Research Design

From Figure 1, it can be seen that there are three stages in pressure vessel design, namely, pressure vessel design, pressure vessel modeling, and stress analysis on pressure vessels. The pressure vessel made is a horizontal type pressure vessel.



Source: Processed Data, 2023

Figure 1. Research Flow Chart

2.2. Data Collection Procedure

a) Literature

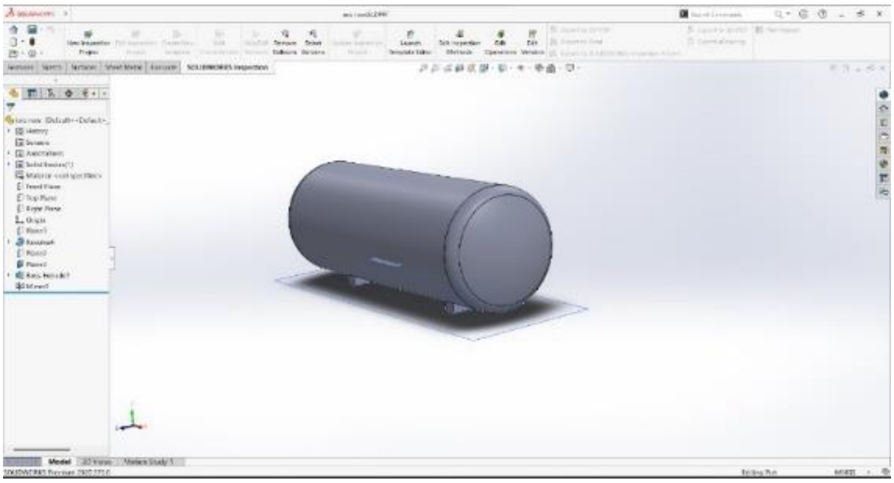
Literature is studied from articles, books, journals related to research, namely stress analysis in pressure vessels using the finite element method.

b) Data Input

The vessel material data used in this study is SS 400 steel. This material data can be entered by opening the Material window and selecting SS 400 steel.

c) Making Pressure Vessel Models with Software ANSYS Student Edition Design Modeler

Modeling geometry vessel pressure using ANSYS Student Edition Design Modeler and then the pressure vessel is made in several components. After all the components are made, the components will be combined into one complete component to be analyzed using ANSYS Student Edition. The components are made solidly and imported into the analysis software with the specified boundary conditions. Pressure vessel models can be created using the ANSYS Student Edition Design Modeler feature . The pressure vessel model used in this tutorial is a horizontal pressure vessel with a diameter of 200 mm, a height of 600 mm, and a thickness of 100 mm. 3 mm wall.



Source: Data Processed with ANSYS Student Edition, 2023

Figure 2. Overall Pressure Vessel Modeling

Figure 2. shows that the pressure vessel has been modeled as a whole and its components have been assembled. The modeling of the pressure vessel is made solid because there is a very significant difference between the thickness and height or other dimensions of the pressure vessel, in addition there is also a thickening in certain parts that make it more possible to model the pressure vessel solidly. This modeling is important because it will determine the analysis that will be carried out next.

a) *Meshing*

Meshing is the process of dividing a pressure vessel model into small elements. Meshing is needed to improve the accuracy of stress analysis results. In this study, meshing with a medium fineness level was used.

b) *Implementation Loading*

After the vessel model press made and meshing is completed , the next step is to apply the loading. The loadings applied to the pressure vessel are as follows: 0.1 MPa; 0.5 MPa; 1 MPa; 1.5 MPa; 2 MPa; 2.5 MPa; 3 MPa.

c) *Analysis Voltage*

After loading applied , the next step is to perform stress analysis. Stress analysis can be performed using the Static Structural feature .

d) *Analysis Results Voltage*

After analysis finished, result analysis voltage can seen in the Results window . The important stress analysis result to note is the maximum stress value. The maximum stress value indicates the strength vessel press to the burden given .

3. RESULTS AND DISCUSSION

3.1. Modeling Pressure Vessel with Ansys Student Edition

In this study, the horizontal pressure vessel structure was modeled according to the standard using the safe ASME section VIII standard. The pressure vessel structure modeling used ANSYS Student Edition Design Modeler software according to the dimensions and geometric shapes in the data. After the geometry was completed, the tubular connection model was converted into ANSYS Student Edition to determine whether the geometric shape was in accordance with the actual pressure vessel structure. The following is the geometric data of the horizontal pressure vessel structure presented in table 1.

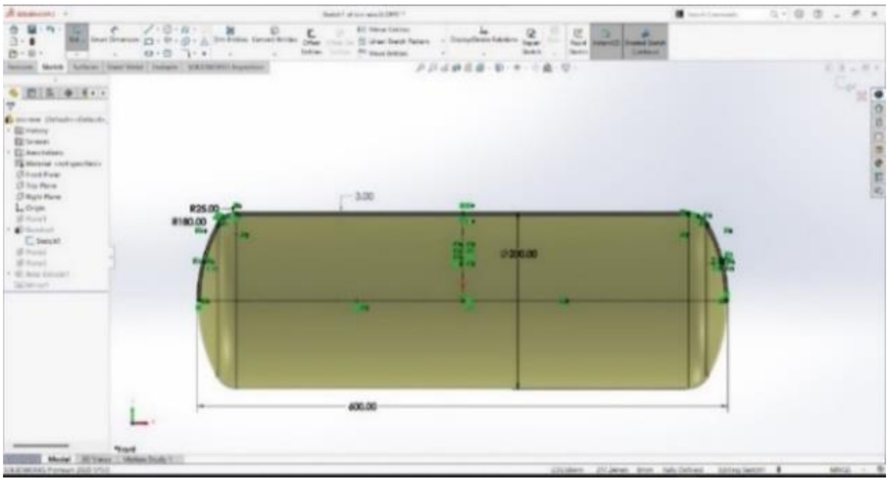
Table 1. Geometric Dimensions of Horizontal Pressure Vessels Structures

Material	SS 400
Outer Diameter	200 mm
Inner Diameter	194 mm
Total Length	600 mm
Thickness	3 mm

Deformasi

Source: Processed Data, 2023

The pressure vessel structure using the ANSYS Student Edition Design Modeler software are shown in Figure 3.



Source: Data processed with ANSYS Student Edition, 2023

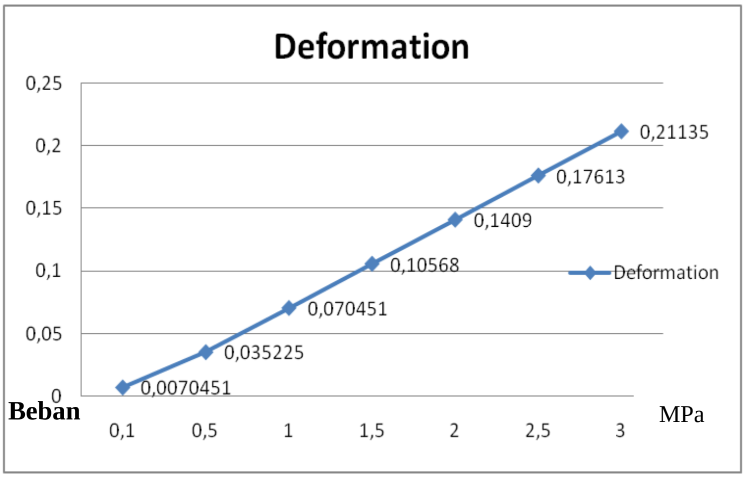
Figure 3. Geometric Dimensions of Horizontal Pressure Vessels Structures

3.2. Deformation, Strain and Stress Analysis on Pressure Vessels

Pressure, deformation, strain, and stress are four interrelated parameters in the analysis of material strength. Pressure is the force per unit area acting on a material. Deformation is the change in shape that occurs in a material due to the pressure acting on it. Strain is the ratio between deformation and the initial dimensions of the material. Stress is the force per unit area acting on a material and causing deformation.

1. Deformation

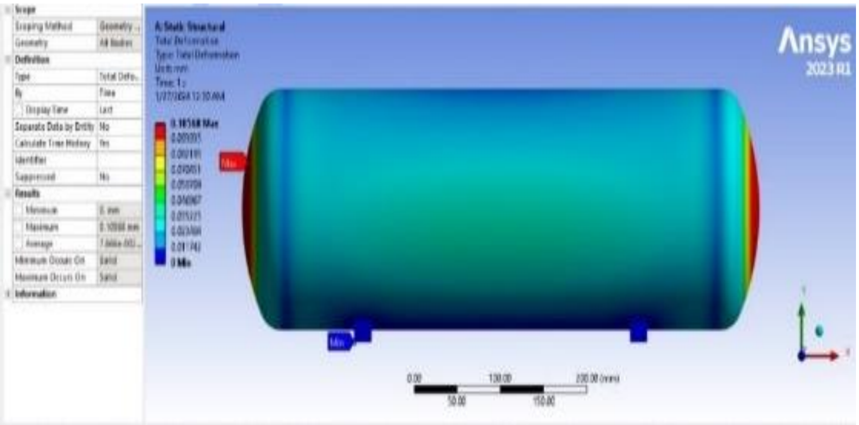
In the context of structural analysis, deformation refers to the change in shape or dimensions of an object in response to a load or force acting on it. Deformation can occur in various forms, including changes in length, changes in angle, or other dimensional changes, depending on the type of load applied and the material properties of the object.



Source: Data processed with ANSYS Student Edition, 2023

Figure 4. Graphical Results of Simulation on Total Deformation of Pressure Vessel

Based on Figure 4, deformation increases with increasing load level. At a load level of 0.1 MPa, the deformation is 0.0070451. At a load level of 3 MPa, the deformation increases to 0.21135. The average deformation is 0.1066.



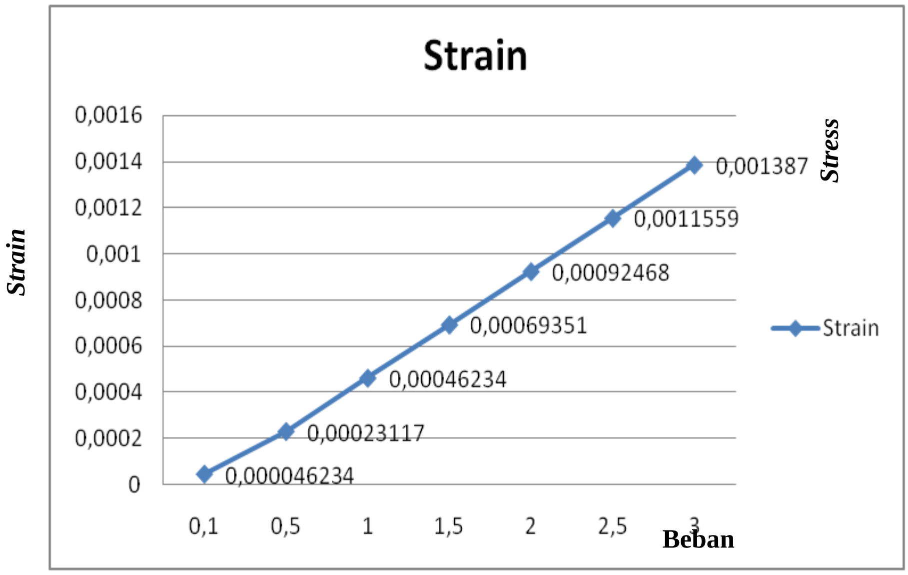
Source: Data processed with ANSYS Student Edition, 2023

Figure 5. Lifting Load Deformation on Pressure Vessels

Figure 5 shows that the maximum stress occurs at the head , with a load of 1.5 MPa and a maximum deformation of 0.10568 mm.

2. Strains

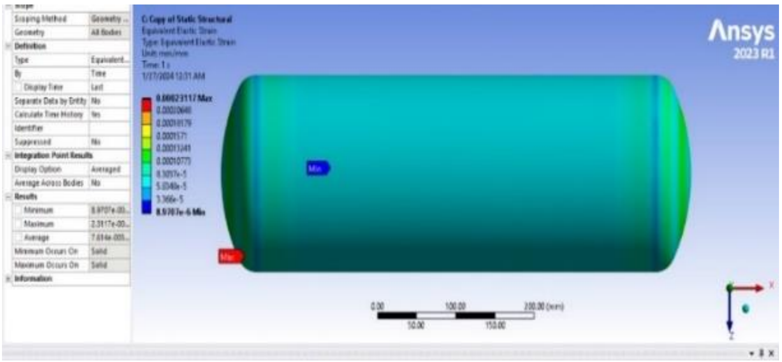
Strain is a measure of the strain that occurs in a material due to an applied load or pressure. In structural analysis, it is important to understand how much a material can stretch before reaching an acceptable strain limit. Based on Figure 6, it is known that the higher the load, the higher the strain. This is due to the compressive force acting on the material. This compressive force causes the material to deform, which ultimately causes strain. At a load of 3 MPa, the strain value reaches 0.001387. This value indicates that the material has undergone significant deformation. If the load continues to be increased, the material will experience even greater deformation, and eventually the material can break.



Source: Data processed with ANSYS Student Edition , 2023

Figure 6.1 Graphical Results of Simulation on Total Strain of Pressure Vessel

Based on Figure 7, it is known that the maximum stress result of the pressure vessel is 0.00023117 MPa at a loading of 0.5 MPa. This value is obtained from the results of the Equivalent Stress simulation , which is a combined stress of normal stress and shear stress.



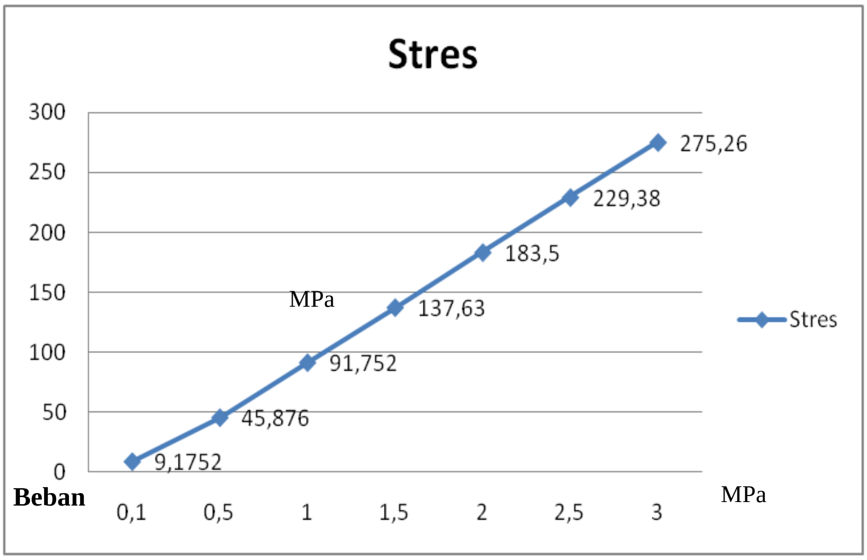
Source: Data processed with ANSYS Student Edition , 2023

Figure 8. Equivalent Elastic Strain in Pressure Vessels

This maximum stress occurs at the bottom of the pressure vessel, namely at the part that is in direct contact with the fluid. This is caused by the presence of a compressive force acting on the bottom of the pressure vessel.

3. Stress

Stress is an important parameter in structural analysis because it provides information about the level of load or force that a material can withstand before reaching the failure limit. Stress calculations help us to evaluate structural safety, determine the appropriate dimensions and materials for an application, and ensure that the material does not exceed the allowable strength limits according to applicable standards and specifications. Stress evaluation is crucial in the design and testing of structures so that they can perform safely and reliably under various operational conditions.

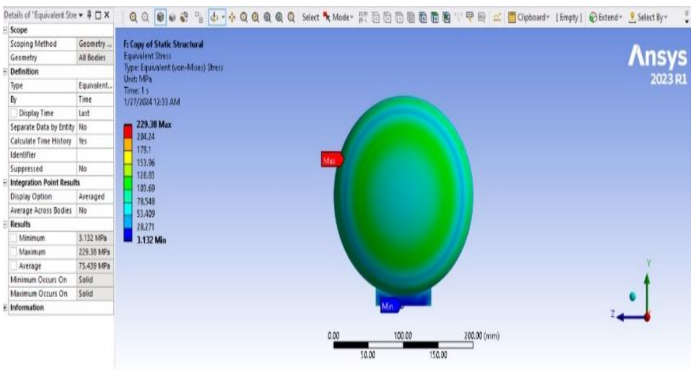


Source: Data processed with ANSYS Student Edition , 2023

Figure 9. Graphical Results of Simulation on Total Stress of Pressure Vessels

Based on the analysis results in Figure 11, it can be concluded that there is a linear relationship between pressure and stress in the material or structure. This means that every 1 MPa increase in pressure will cause an increase in stress of 91.752 MPa. An increase in pressure of 0.5 MPa (from 0.5 MPa to 1 MPa) causes an increase in stress of 45.876 MPa and so on. This shows consistency in the material's response to increasing pressure that continues proportionally.

Figure 10 shows that the highest stress on the pressure vessel wall is 229.38 psi at a loading of 2.5 MPa. Thus, the maximum stress that occurs is smaller than the allowable stress of the material used, which is 235 MPa, so the pressure vessel is safe.



Source: Data processed with ANSYS Student Edition, 2023

Figure 102. Equivalent (Von-Mises) Stress in Pressure Vessels

In the stress- strain analysis for pressure vessels with loads of 0.1 to 3 MPa, deformation, strain, and stress data were obtained as shown in Table 3. As a reference, the allowable stress (yield strength) is 235 MPa.

Table 3. Structural Static Test Result Data

MPa	Deformation (mm)	Strains	Stress (MPa)
0.1	0.0070451	0.000046234	9,1752
0.5	0.035225	0.00023117	45,876
1	0.070451	0.00046234	91,752
1.5	0.10568	0.00069351	137.63
2	0.1409	0.00092468	183.5
2.5	0.17613	0.0011559	229.38
3	0.21135	0.001387	275.26

Source: Data processed with ANSYS Student Edition , 2023

It can be seen that at each loading point, the stress value generated is still far below the specified allowable stress (235 MPa). For example, at a load of 2 MPa, the stress generated is 183.5 MPa, which is still far below the allowable stress. However, the stress value for a load of 3 MPa is 275.26 MPa. This means that it exceeds the allowable stress value of 235 MPa that has been set. Therefore, based on these data, at a load of 3 MPa, this pressure vessel does not meet the specified allowable stress limits.

In practice, if the stress value exceeds the allowable stress, it may indicate that the pressure vessel is in an unsafe condition and may experience permanent deformation or even structural failure. Design review or material changes are required to ensure the safety and reliability of the pressure vessel at a load of 3 MPa.

4. Safety Factor (SF)

Safety factor (SF) is a factor used to account for uncertainty factors in the design of a structure or component. This factor is used to ensure that the structure or component can withstand the given load without being damaged.

Table 4. Data on the Relationship between Safety Factor and Yield Strength

Safety Factor	Yield Strength
1	235
1.5	156
2	117.5
2.5	94
3	78

Source: Data processed with ANSYS Student Edition , 2023

The safety factor is the most important benchmark used in determining the quality of a design. The reference is if the factor of safety (FOS) value is at least less than 1 then the design is of poor quality and not suitable for use and tends to be dangerous, conversely if the FOS value is more than 1 and usually between 1-3 then the design is of good quality, safe and suitable for use, the safety factor also indicates the level of ability of a material to receive external loads in this case is the compressive load. If the FOS value reaches at least 3 digits or more, then the design is safe, of good quality but the price is very expensive and tends to have a very large weight. Most Pressure Vessels have a factor of safety of 2. In the analysis of this horizontal pressure vessel, the safety value is obtained as shown in table 4.

From table 4, it can be seen that there is a relationship between Safety Factor (SF) and Yield Strength (YS). Here are some interpretations that can be made:

1. Inverse Relationship: There is an inverse relationship between SF and YS. The higher the SF, the lower the allowable YS. This indicates that at a higher safety level (high SF), materials with lower yield strength can be used. Conversely, at a lower safety level (low SF), materials with higher yield strength can be used . tall needed .
2. Threshold Limit Value: In this study, the allowable stress is 235 MPa. This means that the structure or component must be designed in such a way that the stress it experiences does not exceeding 235 MPa.

From the overall description above, it can be concluded that the pressure vessel analyzed has met the required criteria and is feasible to be produced at static loading up to 2.5 MPa, where the test object experiences deformation of about 0.17613 mm with a strain of about 0.0011559, and a stress of about 229.38 MPa. Thus, the maximum stress experienced by this pressure vessel is 229.38 MPa. At a loading of 3 MPa, the stress has exceeded the material's allowable stress of 235 MPa. The excessive stress on the pressure vessel occurs at the base of the pressure vessel which is in direct contact with the fluid. This is caused by the presence of force press that works on the part.

4. CLOSING

Based on results research that has been done, then the tension that occurs can be withdrawn conclusion study This refers to the ASME Section VIII standard . The results of the analysis with ANSYS Student Edition shows that with increasing load on the pressure vessel, there is an increase in deformation, strain, and stress. In general, at higher load levels, deformation, strain, and stress also increase. At a loading of 0.1 MPa, the deformation is 0.0070451 mm with a strain of 0.000046234 and a stress of 9.1752 MPa; at 0.5 MPa, the deformation reaches 0.035225 mm with a strain of 0.00023117 and a stress of 45.876 MPa; at 1 MPa, the deformation is 0.070451 mm with a strain of 0.00046234 and a stress of 91.752 MPa; at 1.5 MPa, the deformation reaches 0.10568 mm with a strain of 0.00069351 and a stress of 137.63 MPa; at 2 MPa, the deformation is 0.1409 mm with a strain of 0.00092468 and a stress of 183.5 MPa; at 2.5 MPa, the deformation reaches 0.17613 mm with a strain of 0.0011559 and a stress of 229.38 MPa; finally, at 3 MPa loading, the deformation is 0.21135 mm with a strain of 0.001387 and a stress of 275.26 MPa.

The pressure vessel analyzed has met the required criteria and is feasible to be manufactured at static loading up to 2.5 MPa, where the test object experiences a deformation of approximately 0.17613 mm with a strain of approximately 0.0011559, and a stress of approximately 229.38 MPa. Thus, the maximum stress experienced by the vessel press This of 229.38 MPa. Stress excessive pressure on pressure vessels occurs at the bottom of the pressure vessel which is in direct contact with the fluid. This is caused by the presence of force press that works on the part.

BIBLIOGRAPHY

Aziz, A., Hamid, A., & Hidayat, I. (2014). Pressure Vessel Design for Separation 3. SINERGI , 18(1) , 31–38. https://doi.org/https://dx.doi.org/10.22441/sinergi

Bhafikatti, S.S. (2015). Finite Element Analysis. New Age International Publishers , Third Edit .

Effendi, R. (2014). Horizontal Strength Optimization of Vessel Using Finite Element Analysis. Proceedings of Semnastek , 1(1).

Huda, K., & Permadi, LC (2017). Compressive Strength Analysis of Natural Gas (CNG) Cylinders Using Finite Element Method. Proceedings of SENIATI , 3(2), E19-1.

Kumar, V., Kumar, N., Angra, S., & Sharma, P. (2014). Design of Saddle Support for Horizontal Pressure Vessel. International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering , 8(12) , 1–5.

Manullang, E., Tangkuman, S., & Maluegha, B. (2016). Stress Analysis on Vertical Pressure Vessel 13ZL100040291 at PT. Aneka Gas Industri. JOURNAL OF POROS MECHANICAL ENGINEERING UNSRAT , 5(2) .

Moss, DR (2013). Basic, Moss, DR Pressure Vessel Design Manual (4th Edition). In Elsevier Butterworth Heinemann.

Nakasone, Stolarski, & Yoshimoto. (2018). Engineering Analysis With ANSYS Software . Elsevier Butterworth-Heinemann.

Nicholasta, K.L., Kurdi, O., & Satrijo, D. (2023). Stress Analysis on Industrial-Scale Pressure Vessels Using Finite Element Method. Journal of Mechanical Engineering, 11(2), 123-124.

Purnomo, J., & Satrijo, DI (2012). Design and Analysis of Stress in Vertical Pressure Vessels Using Finite Element Method . Diponegoro University.

Ramadhan, R., Risano, AYE, & Su'udi, A. (2013). Thermal and stress analysis in the design of pressure vessels for palm oil waste with a capacity of 10,000 tons/month. Scientific Journal of Mechanical Engineering , 1(4) .

Santoso, DR (2017). Mechanical Stress Measurement Based on Piezoelectric Sensor: Design Principle and Implementation . Universitas Brawijaya Press.

Satrijo, D., & Habsya, SA (2012). Design and analysis of stress in horizontal pressure vessels using the finite element method. Rotation , 14(3) , 32–40.

Versteeg, H.K., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: the finite volume method . Pearson education.