IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 23, Issue 3 Ser. 1 (May - June 2026), PP 24-30
www.iosrjournals.org

Stress Analysis On Welded Joints Of A Storage Tank Using Solidworks 2020

M. Andri Aditya Almahesa[1], Ahmad Zamheri[2], Rizky Yuliandi[2], Aneka Firdaus[1], Amrifan Saladin Mohruni[1], Jimmy Dn[1], Hendri Chandra[1], Agung Mataram[1]
(Department Of Mechanical Engineering, Universitas Sriwijaya, South Sumatra, Indonesia)
(Department Of Mechanical Engineering, Sriwijaya State Polytechnic, South Sumatra, Indonesia)

Abstract:
This study analyzes stress distribution, strain, and displacement in welded joints of a fixed cone roof storage tank using the finite element method (FEM) with SolidWorks 2020. The tank design follows API 650 standards and uses ASTM A283 steel material. The objective is to identify critical stress regions and evaluate the structural safety of the tank under internal pressure loading. A numerical simulation was performed by applying an internal pressure of 0.08387 MPa to the tank model. The results show that the maximum Von Mises stress is 74.73 MPa, which is significantly lower than the material yield strength, indicating that the structure remains within the elastic region and is considered safe under operating conditions. The highest stress concentration occurs in the welded joint area, which is identified as the critical region of the structure. In addition, the maximum displacement obtained from the simulation is 4.027 mm, which is still within acceptable limits. The findings demonstrate that the application of FEM using SolidWorks provides an effective and reliable approach for evaluating the mechanical behavior and structural integrity of welded storage tanks.

Keywords: *Stress analysis, welded joint, storage tank, finite element method, Von Mises, SolidWorks*

Date of Submission: 01-05-2026 Date of Acceptance: 11-05-2026

I. Introduction

Storage tanks play a crucial role in the oil, gas, and chemical industries as they are used to store large volumes of fluids under specific pressure conditions. One of the most commonly used types is the fixed cone roof tank, which is designed to store fluids with relatively low vapor pressure. During operation, the tank is subjected to internal pressure that induces stress, strain, and deformation within the structure, particularly at welded joints. Therefore, structural integrity analysis is essential to ensure the safety and reliability of the tank throughout its service life.

Previous studies have investigated stress analysis in pressure vessels and storage tanks using numerical approaches. Aziz et al. (2014) highlighted the importance of proper pressure vessel design by considering stress distribution due to internal pressure. Chandra and Saputra (2016) employed finite element-based software to analyze stress in LPG pressure vessels and demonstrated that numerical methods can provide accurate results compared to analytical calculations. Furthermore, Logan (2012) explained that the Finite Element Method (FEM) is an effective numerical technique for solving complex mechanical problems, including stress, displacement, and deformation analysis. However, most of these studies focus on general pressure vessel structures and do not specifically address stress distribution in welded joints of fixed cone roof tanks.

Based on the aforementioned studies, the scientific novelty of this research lies in a focused analysis of stress, strain, and displacement distribution in welded joints of a fixed cone roof tank using FEM-based simulation with SolidWorks 2020. This study not only evaluates the maximum stress but also identifies critical regions that are prone to structural failure.

The research problem addressed in this study is how stress, strain, and displacement are distributed in the welded joints of a fixed cone roof tank under internal pressure loading, and whether the structure remains within safe limits based on the material yield strength criterion.

The objective of this study is to analyze the distribution of stress, strain, and displacement in welded joints of a fixed cone roof tank using the finite element method, as well as to identify critical regions and evaluate the structural safety based on simulation results.

II. Research Method

This study employs a numerical simulation approach using the Finite Element Method (FEM) to analyze stress, strain, and displacement in a fixed cone roof storage tank. The analysis is carried out using

SolidWorks 2020 as the primary simulation tool. The research begins with data collection based on field observations, literature review, and documentation of tank specifications. The tank design follows API 650 standards and uses ASTM A283 steel as the material. The main dimensions include a diameter of 36.46 m and a height of 11.7 m, with varying shell thicknesses.

The geometric model of the fixed cone roof tank is developed using SolidWorks 2020 based on the actual design specifications. The model is constructed to accurately represent the structural configuration of the tank. The resulting geometry is shown in Figure 1.

Figure 1 Geometric model of the fixed cone roof tank developed in SolidWorks

Material properties are then assigned to the model to simulate its mechanical behavior. The material used is ASTM A283 steel, with a Young's modulus of 200 GPa, Poisson's ratio of 0.29, density of 7800 kg/m³, and yield strength of 260 MPa.

Boundary conditions are applied to represent the actual constraints of the tank during operation. The bottom of the tank is defined as a fixed support to prevent displacement and ensure structural stability. The applied boundary condition is illustrated in Figure 2.

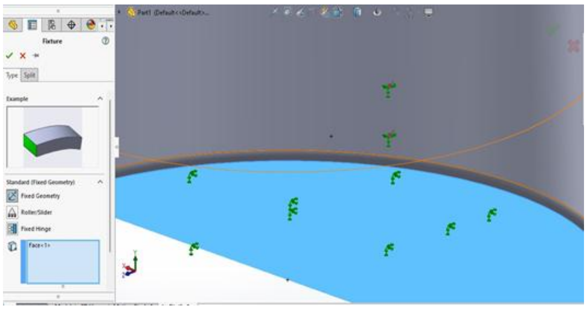


Figure 2 Fixed support applied at the bottom of the tank

The loading conditions are applied in the form of internal and external pressures. An internal pressure of 0.08387 MPa is uniformly distributed on the inner surface of the tank to simulate operational loading. This condition is presented in Figure 3.

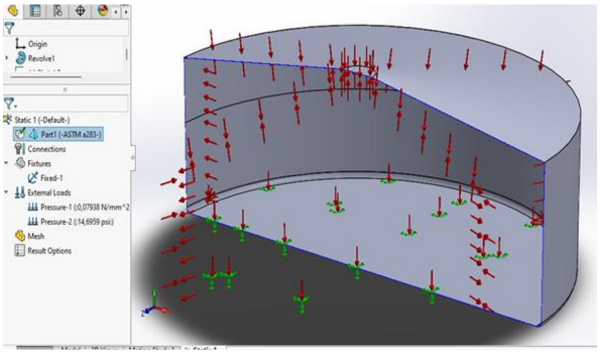


Figure 3 Internal pressure applied to the inner surface of the tank

Additionally, external pressure is applied to the outer surface of the tank to represent environmental effects and realistic working conditions. The applied external pressure is shown in Figure 4.

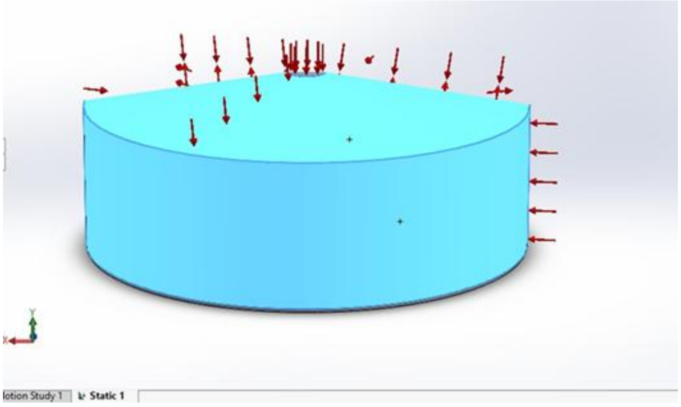
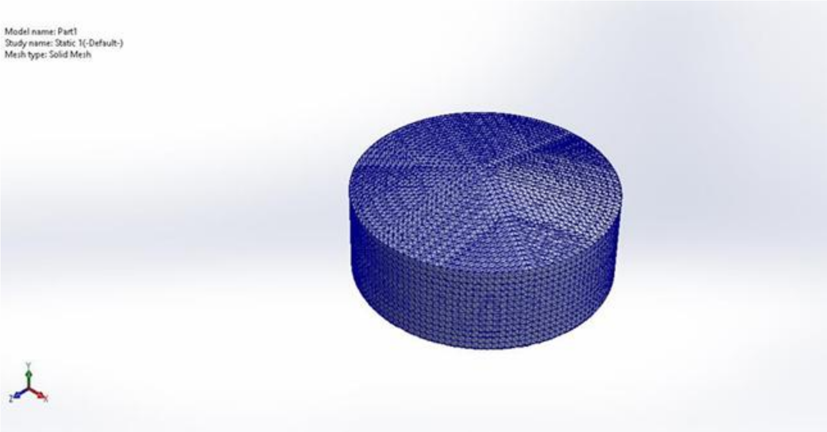


Figure 4 External pressure applied to the outer surface of the tank

The model is then discretized using a meshing process to enable numerical analysis. A solid mesh with high quality is used to ensure accurate stress distribution, particularly in critical areas such as welded joints. The mesh consists of approximately 61,420 nodes and 31,981 elements. The meshing result is shown in Figure 5.



Model name: Part1
Study name: Static 1(-Default-)
Mesh type: Solid Mesh

Figure 5 Finite element mesh of the tank model

After meshing, the simulation is executed to obtain the distribution of stress, strain, and displacement. The analysis focuses on the Von Mises stress as the primary parameter for evaluating structural safety. The simulation results are further analyzed to identify critical regions and to determine whether the stress values remain below the material yield strength.

III. Result And Discussion

The simulation results obtained from the finite element analysis provide detailed information on the distribution of stress, strain, and displacement in the fixed cone roof tank structure. The analysis primarily focuses on identifying critical regions and evaluating structural safety based on the Von Mises stress criterion.

Stress Distribution (Von Mises Stress)

The results indicate that the maximum Von Mises stress reaches 74.73 MPa, which is significantly lower than the material yield strength. This finding suggests that the tank structure remains within the elastic region and is structurally safe under the applied loading conditions. From an engineering standpoint, this confirms that the tank design satisfies the strength requirement and maintains structural integrity during operation. The distribution of Von Mises stress obtained from the simulation is shown in Figure 6.

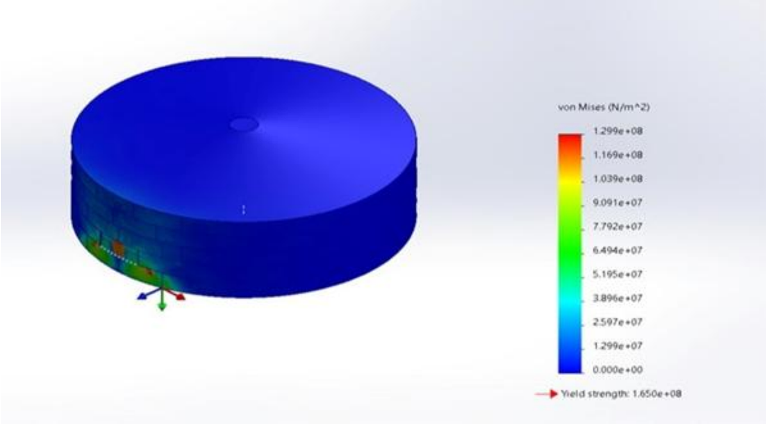


Figure 6 Von Mises stress distribution of the tank model

A more detailed observation of the stress contour reveals that the maximum stress is concentrated in the welded joint regions. This phenomenon can be explained by the presence of geometric discontinuities and material heterogeneity introduced during the welding process. In welded structures, abrupt changes in geometry—such as thickness transitions, joint angles, and weld profiles result in localized stress amplification. According to the theory of stress concentration, any discontinuity in a structure disrupts the uniform distribution of stress, leading to higher stress intensity in specific regions.

Furthermore, residual stresses generated during the welding process may also contribute to the increase in local stress levels. Thermal cycles during welding cause uneven expansion and contraction, which can induce additional internal stresses even before external loads are applied. When combined with operational pressure, these residual stresses may intensify the overall stress state in the welded area.

From a theoretical perspective, the stress distribution in the tank shell is primarily governed by thin-walled pressure vessel theory. The internal pressure acting on the cylindrical shell generates two principal stresses: circumferential (hoop) stress and longitudinal stress. The hoop stress can be expressed as:

$$\sigma_h = \frac{P \cdot D}{2t}$$

$$\sigma_l = \frac{P \cdot D}{4t}$$

where P is the internal pressure, D is the tank diameter, and t is the shell thickness. These equations indicate that the hoop stress is theoretically twice the longitudinal stress, which explains why higher stress values are predominantly observed along the circumferential direction. This analytical behavior is consistent with the simulation results, confirming the validity of the FEM model.

In addition, the variation in shell thickness across different rings of the tank also influences the stress distribution. Thinner sections tend to experience higher stress due to reduced load-carrying capacity, while thicker sections provide greater مقاومت (resistance) to deformation. This gradual thickness variation helps distribute stress more efficiently, but localized effects at welded joints remain significant.

When compared to previous studies, the present findings are in good agreement with established research. Chandra and Saputra (2016) reported that stress concentration in pressure vessels is commonly found near welded joints and geometric discontinuities. Similarly, Logan (2012) emphasized that finite element analysis is particularly effective in capturing localized stress concentrations that cannot be accurately predicted using simplified analytical methods. The consistency between the present results and previous studies reinforces the reliability of the simulation approach.

Moreover, the ratio between the maximum Von Mises stress (74.73 MPa) and the material yield strength indicates a substantial safety margin. This margin is critical in engineering design, as it accounts for uncertainties such as material imperfections, loading fluctuations, and manufacturing defects. Therefore, the obtained stress level suggests that the structure not only meets the minimum safety requirements but also possesses a reasonable factor of safety.(Manullang et al., 2007)

Based on these results, the primary scientific finding of this study is that the welded joint region constitutes the most critical area in the tank structure due to stress concentration effects. However, despite this localized increase in stress, the overall stress level remains well below the allowable limit, indicating that the structure is safe under the given operating conditions.

Strain Distribution

The results show that the strain distribution follows a similar pattern to the stress distribution, with higher strain values observed in regions experiencing higher stress. This is consistent with Hooke's law, which states that strain is directly proportional to stress within the elastic region of the material. The strain distribution obtained from the simulation is presented in Figure 7.

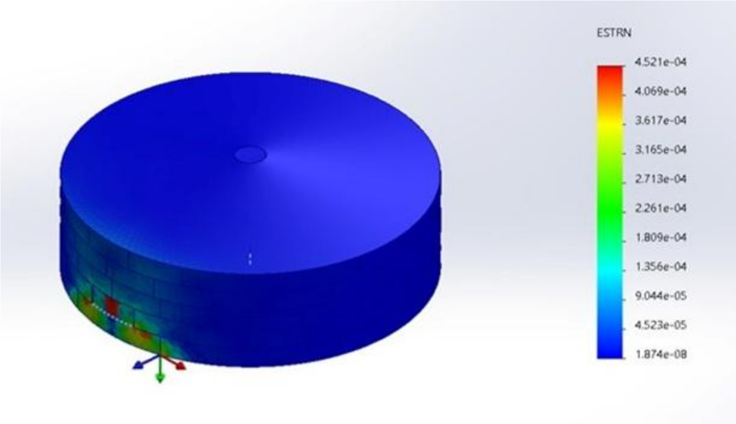


Figure 7 Strain distribution of the tank mode

The relatively low strain values indicate that the deformation experienced by the structure remains within the elastic limit, meaning that the material is able to fully recover its original shape once the applied load is removed. This behavior confirms that no plastic deformation occurs under the given loading conditions, which is a key requirement for maintaining long-term structural integrity in storage tank applications. (Taler et al., 1999)

From a mechanical standpoint, the strain distribution is strongly influenced by both material stiffness and structural geometry. According to Hooke's law, strain is inversely proportional to the modulus of elasticity; therefore, materials with higher stiffness tend to exhibit lower strain under the same loading conditions. In this case, the use of ASTM A283 steel, which has relatively high elastic modulus, contributes to limiting the overall deformation of the structure.

In addition, geometric factors such as shell thickness variation play a significant role in strain behavior. Regions with thinner shell thickness exhibit relatively higher strain due to reduced load-carrying capacity and stiffness. This is consistent with classical shell theory, where strain is more pronounced in areas with lower structural rigidity. However, the gradual variation in shell thickness across the tank helps to distribute deformation more evenly and prevents excessive strain localization. (Satrijo & Habsya, 2012)

It is also important to note that the presence of welded joints contributes to localized strain concentration. Similar to stress behavior, strain tends to increase in regions where geometric discontinuities exist. The welded areas act as transition zones where load transfer occurs, leading to localized deformation. Nevertheless, the magnitude of strain in these regions remains within acceptable limits, indicating that the welded joints are still structurally reliable under the applied loading.

These findings are consistent with the general behavior of thin-walled pressure vessels, where deformation is typically small and relatively uniform, except in areas affected by discontinuities or stress concentration. Previous studies have also reported similar trends, where strain distribution closely follows stress patterns and remains within elastic limits under normal operating conditions.

Overall, the scientific finding from this analysis is that the strain distribution is well-controlled and does not indicate any risk of excessive deformation or structural instability. The results further confirm that the tank design is adequate in maintaining elastic behavior under the specified loading conditions. (Moss, 2004)

Displacement Analysis

The maximum displacement obtained from the simulation is 4.027 mm, which occurs at the upper section of the tank structure. This displacement is relatively small compared to the overall dimensions of the tank, indicating good structural rigidity. The displacement distribution of the tank model is shown in Figure 8.

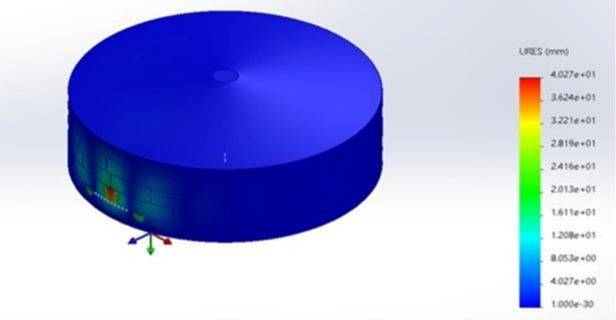


Figure 8 Displacement distribution of the tank mode

The displacement behavior can be explained by the effect of internal pressure acting on the tank walls, which induces outward expansion of the cylindrical shell. This expansion results in radial displacement that varies along the height of the tank. The upper region of the tank tends to experience higher displacement due to reduced structural constraint compared to the bottom section, which is fixed. This variation is consistent with the boundary condition applied in the model, where the base is fully restrained while the upper structure remains relatively free to deform.

From a structural mechanics perspective, displacement is governed by the relationship between applied load, material stiffness, and boundary conditions. Structures with higher stiffness generally exhibit lower displacement under the same loading conditions. In this study, the use of ASTM A283 steel, combined with the shell geometry, provides sufficient rigidity to limit excessive deformation. However, due to the large diameter-to-thickness ratio characteristic of thin-walled structures, some degree of radial expansion is inevitable. (Matthews, 2014)

Furthermore, the displacement distribution is also influenced by the variation in shell thickness along the height of the tank. Sections with thinner walls tend to exhibit slightly higher displacement due to reduced bending stiffness and load resistance. This behavior aligns with classical shell theory, where deformation is inversely related to structural rigidity. Nevertheless, the gradual reduction in thickness is designed to optimize material usage while maintaining acceptable deformation levels. (Khan, 2010)

It is also important to consider the role of welded joints in displacement behavior. Although the primary influence of welded joints is on stress concentration, they can also affect local deformation due to differences in stiffness and material continuity. However, in this case, no significant abnormal displacement is observed in the welded regions, indicating that the joint design does not introduce excessive flexibility into the structure. (Huda & Permadi, 2017)

The obtained maximum displacement of 4.027 mm is relatively small when compared to the overall dimensions of the tank, indicating that the structure maintains good geometric stability. From an engineering perspective, this level of displacement is acceptable and does not pose a risk to structural integrity or operational performance.

Overall, the analysis demonstrates that the displacement behavior of the tank is well within acceptable limits and is consistent with theoretical expectations for thin-walled pressure vessels. This finding confirms that the applied design and boundary conditions provide adequate stiffness and stability under the given loading conditions.

Structural Safety Evaluation

The structural safety of the fixed cone roof tank is evaluated based on the Von Mises failure criterion, which is widely used for ductile materials such as ASTM A283 steel. According to this criterion, yielding occurs when the equivalent Von Mises stress reaches or exceeds the material yield strength.

Based on the simulation results, the maximum Von Mises stress obtained is 74.73 MPa, which is significantly lower than the material yield strength. This indicates that the structure operates well within the elastic region and is not subjected to yielding under the applied loading conditions. From an engineering perspective, this confirms that the tank design satisfies the fundamental strength requirements. (Bednar, 1981)

To further assess structural safety, the factor of safety (FoS) can be considered as the ratio between the material yield strength and the maximum induced stress. A higher FoS indicates a safer design with a greater margin against failure. In this study, the relatively low stress level compared to the yield strength suggests that the structure possesses an adequate safety margin to accommodate uncertainties such as material imperfections, loading variations, and potential degradation over time.

In addition, the identification of welded joints as critical regions provides important insight into the structural behavior of the tank. This indicates that the welding design and fabrication quality are sufficient to withstand the applied loads without causing structural failure. However, from a practical standpoint, these regions should still be prioritized during inspection and maintenance, as they are more susceptible to fatigue and crack initiation under cyclic loading conditions.

Furthermore, the absence of excessive strain and displacement, as discussed in the previous sections, supports the conclusion that the structure maintains both strength and stiffness requirements. The combined evaluation of stress, strain, and displacement provides a comprehensive understanding of the structural performance and confirms that the tank behaves as expected under internal pressure loading.

When compared to design standards such as API 650, the obtained results are consistent with the requirement that operational stresses must remain below allowable stress limits to ensure safe operation. The simulation results demonstrate that the tank meets these criteria, thereby validating the design approach used in this study.

Overall, the main scientific finding from the structural safety evaluation is that the fixed cone roof tank is structurally safe under the specified loading conditions. Despite the presence of stress concentration in welded joints, the stress levels remain within acceptable limits, and no indication of structural failure or instability is observed. This confirms that the design is reliable and suitable for practical application.

The overall structural performance of the tank can be summarized as shown in Table 1.

Parameter	Maximum Value	Allowable Limit / Reference	Status
Von Mises Stress	74.73 MPa	260 MPa (Yield Strength)	Safe
Strain	Within elastic range	Elastic limit of material	Safe
Displacement	4.mm	Within acceptable limit	Safe

Table 1. Structural Performance

IV. Conclusion

This study has successfully analyzed the stress, strain, and displacement distribution in a fixed cone roof storage tank using the Finite Element Method (FEM) with SolidWorks 2020. The results demonstrate that the structural response of the tank under internal pressure loading remains within safe limits, as the induced stress does not exceed the material yield strength. This indicates that the tank operates entirely within the elastic region and satisfies the required strength criteria.

The main scientific finding of this study is that welded joints represent the most critical regions in the tank structure due to stress concentration effects caused by geometric discontinuities. However, despite the localized increase in stress, the overall structural integrity is not compromised, as all evaluated parameters including stress, strain, and displacement remain within acceptable limits. This confirms that the design of the tank is structurally reliable and suitable for operational conditions.

Furthermore, the study demonstrates that the application of FEM provides an effective and accurate approach for evaluating the mechanical behavior of storage tanks, particularly in identifying critical regions that cannot be easily detected through conventional analytical methods.

For future work, it is recommended to extend the analysis by considering additional factors such as dynamic loading, thermal effects, and fatigue behavior in welded joints to provide a more comprehensive assessment of long-term structural performance.

References

[1]. Aziz, A., Hamid, A., & Hidayat, I. (2014). Perancangan Bejana Tekan (Pressure Vessel) Untuk Separasi 3 Fasa. Sinergi, 18(1), 31–38.

[2]. Bednar, H. H. (1981). Pressure Vessel Design Handbook.

[3]. Chandra, H., & Saputra, A. (2016). Analisis Tegangan Pada Bejana Tekan Lpg Kapasitas 3 Kg Dengan Bantuan Software Abaqus 6.14.

[4]. Huda, K., & Permadi, L. C. (2017). Analisis Kekuatan Compresive Natural Gas (Cng) Cylinders Menggunakan Metode Elemen Hingga. Seminar Nasional Inovasi Dan Aplikasi Teknologi, 15.

[5]. Khan, S. M. A. (2010). Stress Distributions In A Horizontal Pressure Vessel And The Saddle Supports. International Journal Of Pressure Vessels And Piping, 87(5), 239–244. Https://Doi.Org/10.1016/J.Ijpvp.2010.03.005

[6]. Logan, D. L. (2013). A First Course In The Finite Element Method, 5th Ed. In Journal Of Chemical Information And Modeling (Vol. 53, Number 9).

[7]. Manullang, E., Tangkuman, S., & Maluegha, B. L. (2007). Analisis Tegangan Pada Bejana Tekan Vertikal 13zl100040291 Di Pt. Aneka Gas Industri. Jurnal Online Poros Teknik Mesin, 5(2), 92–102.

[8]. Matthews, C. (2014). Engineers' Guide To Rotating Equipment: The Pocket Reference. In Engineers' Guide To Rotating Equipment: The Pocket Reference. Https://Doi.Org/10.1002/9781118903100

[9]. Moss, D. R. (2004). Pressure Vessel Design Manual (3rd Ed.). Gulf Professional Publishing. Https://Doi.Org/10.1016/B978-0-7506-7740-0.X5000-8

[10]. Satrijo, D., & Habsya, S. A. (2012). Perancangan Dan Analisa Tegangan Pada Bejana Tekan Horizontal Dengan Metode Elemen Hingga. Rotasi Jurnal Teknik Mesin, 14.

[11]. Taler, J., Węglowski, B., Zima, W., Gradziel, S., & Zborowski, M. (1999). Analysis Of Thermal Stresses In A Boiler Drum During Start-Up. Journal Of Pressure Vessel Technology, Transactions Of The Asme, 121(1). Https://Doi.Org/10.1115/1.2883672