

## Modeling and Static Analysis of Pressure Vessel in Full Welded Connections

Lydia Anggraini<sup>a,c\*</sup>, Rahmat Hidayat Siregar<sup>a</sup>, Ronald Sukwadi<sup>b,c</sup>, Rendi Hernawan<sup>a</sup>

<sup>a</sup> Mechanical Engineering Study Program, Faculty of Engineering, President University  
Jl. Ki Hajar Dewantara, Jababeka Education Park, Cikarang, Bekasi, West Java 17550, Indonesia

<sup>b</sup> Department of Industrial Engineering, Atma Jaya Catholic University of Indonesia  
Jl. Jend. Sudirman No.51, Karet Semanggi, South Jakarta, Jakarta 12930, Indonesia

<sup>c</sup> Professional Engineer Study Program, Atma Jaya Catholic University of Indonesia  
Jl. Jend. Sudirman No.51, Karet Semanggi, South Jakarta, Jakarta 12930, Indonesia

\*E-mail: lydia.anggra@president.ac.id

### Abstract

The pressure analysis on the cylindrical pressure vessel is simulated computationally with the finite element method and evaluated from some point of view performance parameters. In this study, the stress elastic analysis method is used with basic calculations to determine the design thickness of the shell, head, nozzle, and material specifications used for pressure vessel analysis. For contraction design by using Autodesk Inventor software for 3D models. This analysis used the finite element method with ANSYS Workbench software, including thermal transient analysis of welded joints then connected with structural analysis of analysts for construction design. It includes material properties and weld joint design on each part of the overall pressure vessel. This result can be used to more rapidly identify critical design variables to optimize the design and find out the maximum deformation occurring in each part of the pressure vessel. The resulting analysis leads to the selection of appropriate measurements for structural use and thermal transient heat transfer analysis on the weld joints on the adjacent welding Nozzle. Interconnect and overlap welded joints with manhole pad welding and identification of the most critical deformation measurements based on parameters.

**Kata kunci:** modeling; pressure vessel; static analysis; welding

### Abstrak

Analisis tekanan pada bejana tekan silinder disimulasikan secara komputasi dengan metode elemen hingga dan dievaluasi dari beberapa sudut pandang parameter kinerja. Pada penelitian ini digunakan metode stress elastic analysis dengan perhitungan dasar untuk menentukan ketebalan desain shell, head, nozzle, dan spesifikasi material yang digunakan untuk analisis bejana tekan. Untuk desain kontraksi dengan menggunakan software Autodesk Inventor untuk model 3D. Analisis ini menggunakan metode elemen hingga dengan perangkat lunak ANSYS Workbench, meliputi analisis transien termal sambungan las kemudian dihubungkan dengan analisis struktur analisis untuk desain konstruksi. Ini mencakup sifat material dan desain sambungan las pada setiap bagian dari bejana tekan keseluruhan. Hasil ini dapat digunakan untuk lebih cepat mengidentifikasi variabel desain kritis untuk mengoptimalkan desain dan mengetahui deformasi maksimum yang terjadi di setiap bagian bejana tekan. Analisis yang dihasilkan mengarah pada pemilihan pengukuran yang sesuai untuk penggunaan struktural dan analisis perpindahan panas transien termal pada sambungan las pada Nosel las yang berdekatan. Sambungan las interkoneksi dan tumpang tindih dengan las manhole pad dan identifikasi pengukuran deformasi paling kritis berdasarkan parameter.

**Kata kunci:** analisis statis; bejana tekan; pemodelan; pengelasan

### 1. Introduction

The method of designing pressure vessels in the discussion of this study can be divided into two basic categories, the Design by Calculation (DBC) method and the Design by Analysis (DBA) method [1,2]. Overall dimensions and loads are specified. Wall thickness is calculated by predefined procedures, formulas, and charts. The vessel geometry and overall dimensions are defined. Allowable loads are evaluated through detailed structural analysis [3]. The objective of this study is to analyze the pressure vessel with the modeling and static analysis approach method. Analysis of the structure is divided into elastic and plastic analysis depending on the material characteristics.

Thin wall pressure vessels (TWPV) are widely used in industry for liquids and gases storage and transportation when configured as tanks [4,5]. They also appear as components of aerospace and marine vehicles such as rocket and balloon skins and submarine hulls. Although in the latter case the vessel is externally pressurized, violating one of the assumptions listed below. The walls of an ideal thin-wall pressure vessel act as a membrane. A sphere is an optimal geometry for a closed pressure vessel in the sense of being the most structurally efficient shape. A cylindrical vessel is somewhat less efficient for two reasons, i.e. (1) the wall stresses vary with direction and (2) closure by end caps can

alter significantly the ideal membrane state, requiring additional local reinforcements, as shown in Figure 1. However, the cylindrical shape may be more convenient to fabricate and transport.



Figure 1. Cylindrical Vessels

## 2. Research Methodology

### 2.1 Design Calculation of Pressure Vessel

Simple pressure vessel geometry is required to obtain comparable analysis results using different methods by selecting a three nozzled cylindrical tank [6-8]. The nozzle is selected from the standard components and is attached perpendicular to the vessel's centerline. Nozzles and other manholes will be analyzed using ANSYS geometry [9, 10], the overall pressure vessel will eventually be analyzed with hydrostatic test analyzers to ascertain the magnitude of deformation after receiving design pressure. The design pressure that can be loaded by a pressure vessel is 900 Psi. Therefore we must first calculate the wall thickness and ellipsoidal head corresponding to the calculation method using ASME Code Sec VIII [11]; 2010 before analyzing by using the FEA method with ANSYS Work-bench.

### 2.2 Geometrical Dimension

Overall dimensions, define as 6808 mm length, ID 2743.2 mm as internal diameter without corrosion and thinning allowance, 1,0 mm corrosion allowance, and 1,0 mm material thinning allowance. The wall thickness dimension is, 70 mm is selected as the initial value of the shell thickness and 77 mm as the initial value of the head thickness. In nozzle 1, a thin wall configuration with the size of 16" Sch 80 on class number 600 is as follows: the outer diameter is 442.8 mm, the internal diameter is 400 mm, the wall thickness is 21.4 mm, and the reinforcement pad diameter and thickness is 907 mm and 70 mm, respectively. In nozzle 2, a thin wall configuration with the size of 16" Sch 80 on class number 600 is as follows, the outer diameter is 442.8 mm, the internal diameter is 400 mm, and the wall thickness is 21.4 mm. In nozzle 3, thin wall configuration data with a size of 10" Sch 120 on class number 600 are as follows, the outer diameter is 292.8 mm, the internal diameter is 250 mm, and the wall thickness is 21.4 mm. In nozzle 4, thin wall configuration data with a size of 8" Sch 120 on class number 600 are as follows, the outer diameter is 236.6 mm, the internal diameter is 200 mm, and the wall thickness is 18.3 mm. Furthermore, in a manhole, thin wall configuration data with a size of 24" Thk 18 mm, the outer diameter is 610 mm, the internal diameter is 574 mm, the wall thickness is 18 mm, and the reinforcement pad diameter and thickness is 1092 mm and 70 mm, respectively.

### 2.3 Drawings Design

The detailed dimensions of the Pressure Vessel and wall thickness can be seen in Figure 2. This drawing design obtained thickness and dimensions after design calculation to determine the thickness of the pressure vessel wall by taking into calculation the acceptable pressure load by ASME VIII Div 1 Pressure Vessel Code Standard.

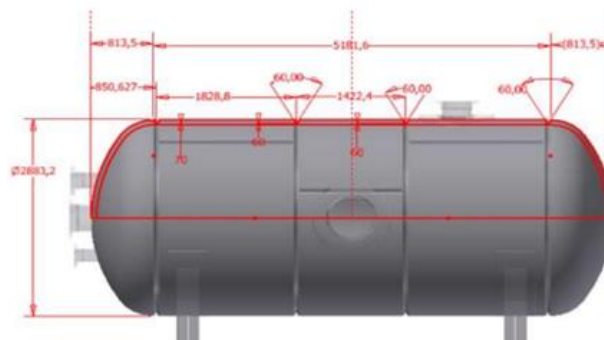


Figure 2. Cylindrical Pressure Vessel Dimension

### 2.4 Material Properties

The specification material selected for the Pressure vessel design includes the Shell section, and the ellipsoidal head is SA-516 Gr.70N. Pipe nozzle using material SA-106 Gr.B and flange SA-105N. And for Saddle support using material type SA-36. The detailed material properties of SA 516 GR 70 N, SA 106 GR. B, SA 105 N, SA 36, E 7016, ER 70S-6, and EH 12 K are described as follows. The modulus of elasticity is 200 GPa, the thermal conductivity of 52 W/m°C, the average density of 7800 kg/m<sup>3</sup>, the coefficient of thermal expansion of 1.2 x 10<sup>-5</sup>, Poisson ratio of 0.3, the bulk modulus of 160 GPa, shear modulus of 76.9 GPa, the average specific heat of 470 J/kg°C. The difference is in the mechanical properties such as tensile strength, yield strength, elongation, and Brinell hardness test value.

### 2.5 Pressure Vessel Load Cases

The amount of load that the pressure vessel has to accept in this analysis is in Table 1.

**Table 1.** Pressure Vessel Load Cases

Description	
Internal operation pressure	900 psi
External pressure	14.7 psi
Maximum allowable working pressure (MAWP)	900 psi
Hydrostatic test pressure	1173 psi
Operating pressure	70 – 160 psi
Internal design temperature	93 °C
Operating normal temperature	27 °C

### 2.6 Acceptance Criteria

To obtain an appropriate value analysis of the analytical methods achieved are given as utilization factors in Equation 1. The analysis result and the corresponding design limit are obtained using the finite element method [12]. The respective design check of the investigated methods is accepted if the ratio is less than one. The final result, including all methods, will be presented in table format for easy comparison.

$$\frac{\text{Analysis Results}}{\text{Design Limit}} \text{ if } (> 1 \text{ fails}) \tag{1}$$

### 2.7 Hydrostatic Test

It is recommended that the metal temperature during the hydrostatic test be maintained at least 17°C above the minimum design metal temperature, but need not exceed 48°C, to minimize the risk of brittle fracture. The test pressure shall not be applied until the vessel and its contents are at about the same temperature. It is recommended to delay the required vessel inspection above if the test temperature exceeds 48°C until the temperature is reduced to 48°C or less. The hydrostatic test pressure at the top of the vessel shall be the minimum of the test pressures calculated by multiplying the basis for calculated-test pressure as defined in 3-2 for each pressure element by 1.3 and reducing this value by the hydrostatic head on that element. The applied test pressure shall be not less than 1.3 times the specified external design pressure. The minimum hydrostatic test pressure shall be the greater of the:

$$P_T = 1.43 \cdot MAWP \tag{2}$$

$$P_T = 1.25 \cdot MAWP \cdot \left( \frac{S_T}{S} \right) \tag{3}$$

Where MAWP is the maximum allowable working pressure, P<sub>T</sub> is the minimum test pressure, S is allowable stress from evaluated at the design temperature, and S<sub>T</sub> is allowable stress from evaluated at the test temperature. In the case of water density, specific weight and thermal expansion coefficient are also considered. Density is the ratio of the mass to the volume of a substance, where ρ is density [g/cm<sup>3</sup>], m is mass [g], and V is volume [cm<sup>3</sup>].

$$\rho = m/V \tag{4}$$

Pure water has the highest density of 1000 kg/m<sup>3</sup>. The specific weight is the ratio of the weight to the volume of a substance defined, where γ is the specific weight, units typically [N/m<sup>3</sup>], m is mass [g], g is the acceleration due to gravity [m/s<sup>2</sup>] which is equal to 9.80665 m/s<sup>2</sup>, V is volume [cm<sup>3</sup>], ρ is density [g/cm<sup>3</sup>].

$$\gamma = (m \cdot g) / V = \rho \cdot g \tag{5}$$

## 3. Results and Discussion

To perform an analysis of the pressure vessel on 3D geometry model must be constructed. The construction design in this study is quite complicated because the whole connection of its part uses welded joints. A 3D model of thin and heavy wall configurations is required. Models should be as accurate to obtain results that approach real structural behavior under imposed loads.

### 3.1 3D Model

The 3D model of the pressure vessel is shown in Fig. For the design of 3D model construction is by using Autodesk Inventor 2017 because each welded joint can be obtained in detail and accurately, as shown in Figure 3. The welded design is carried out between the reinforcement pad and main shell, between the reinforcement pad and the nozzle, and between the nozzle and flange to enhance the model's accuracy. The weld detailed Nozzle is shown in Figure 4 for thin wall pressure vessels is shown in Figure 5.

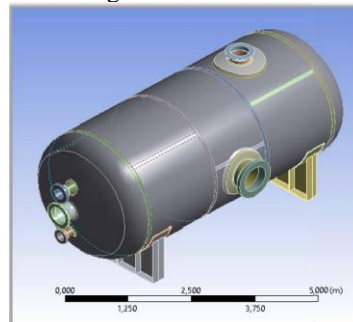


Figure 3. Cylindrical Pressure Vessel 3D Model

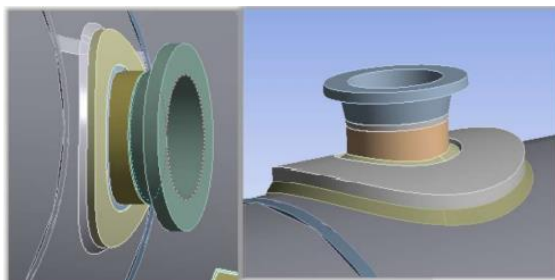


Figure 4. Weld Detail Nozzle 3D Model

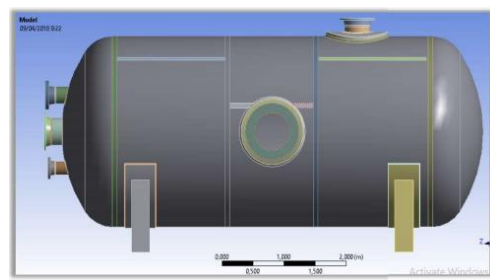


Figure 5. Weld Detail Wall PV 3D Model

In the welded design the front head of the welding joint is directly connected to the nozzle weld joint and then forwarded with the nozzle weld joint to the flange. The weld details Cluster Nozzle in the Head is shown in Figure 6. The cluster weld connection construction of three nozzles on the front head has an elaborate connection and is a very close distance between each nozzle and is not complete with the design of the reinforcement pad. This condition is for the transient thermal welding analysis against deformations arising from areas at very close welding distances of the nozzles. At final, will be simulated stress analysis and hydrostatic test whether this co against the pressure load received by that part. The saddle design supports the static load on the pressure vessel with rib construction on each side. All connections are shown in Figure 7. Saddle support connections include welded fillet weld joints on each side and corner of the plate joint corner. details of saddle support.

These 3D models are then used to create finite element models for further ANSYS Workbench analysis. The process of building up the finite element models is called meshing. For the mesh, the size setting is done trial, and finally got mesh size of 30 mm was according to the dimension and thickness of the wall vessel. The smaller the mesh size used the more accurate the analysis will be. Yet the analysis process in ANSYS software takes a longer time. The initial mesh for the wall press vessel is in Figure 8.

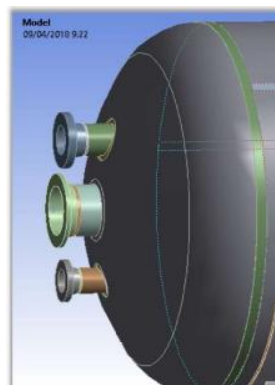


Figure 6. Weld Detail Cluster Nozzle in Head 3D Model

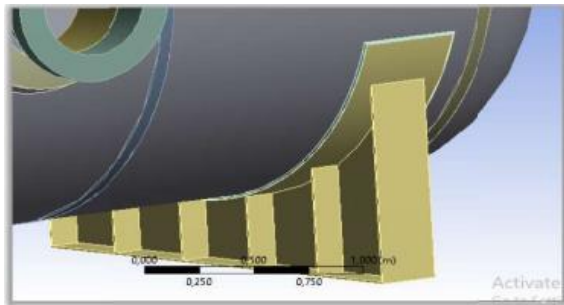


Figure 7. Weld Detail Saddle Support PV 3D Model

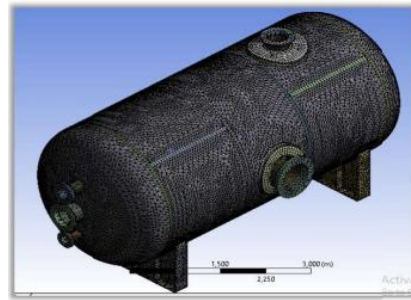


Figure 8. Meshing for Pressure Vessel Wall Geometry

### 3.2 Transient Thermal–Welding Temperature

Transient thermal analysis on three adjacent nozzles is closely-spaced. This condition has sufficient heat load divided into the three cluster nozzle area, which heats the welding temperature of around 1200 °C to 1250 °C for a duration of 7200 seconds until the reduced temperature point to 30°C at the six welding joints. The three joints welding flange connection to the nozzle and the three joints nozzle to the head section.

Look at the intermittent red line showing thermal load analysis occurs due to the weld heat effect. We already know the limit of weld heat flow in the ellipsoidal head area in this area. For this purpose, the result of this thermal analysis is a view and reference for the implementation of welding process activity to preheat before welding. The location is a reference when doing the preheating process.

### 3.3 Static Structural - Equivalent (Von Misses) Stress

The weld heat affected was then linked with the static structural equivalent stress analysis system. So, we obtained the stress analysis result due to the influence of welding heat occurring in the three cluster nozzles area on the Ellipsoidal head dish. The maximum value generated from the static structural equivalent (Von-misses) stress result is 180.86 MPa. This value is still below the yield strength material of Elipsoidal Head SA516 Gr. 70N, where the Yield strength of SA 516 Gr. 70N is 330 MPa. The utilization obtained is  $0.548 < 1$ . Therefore, the analysis result is acceptable.

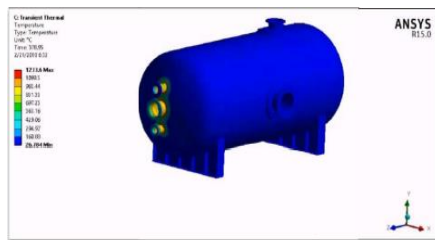


Figure 9. Transient Thermal Load Process of three Cluster Nozzles.

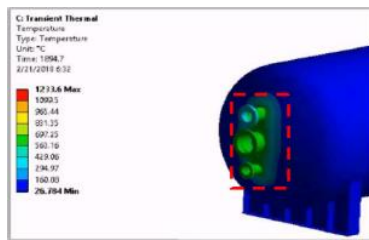


Figure 10. Weld heat affected Location

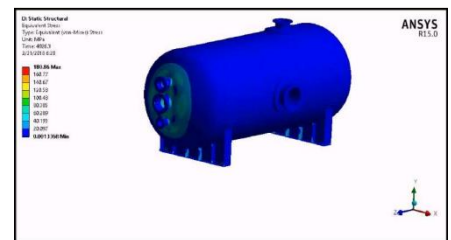


Figure 11. Static structural Equivalent (von-misses) Stress

### 3.4 Static Structural - Total Deformation

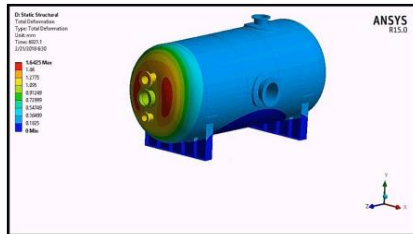
Total deformation is very closely related to the result of weld heat occurring on the head dish, resulting in the elastic deformation of the part. This condition as the basic design reference in analyzing the effect of weld heat on the deformation is still well accepted per the standardization of ASME VIII. The purpose of this design is to get a good and accurate dimension under acceptable measurement tolerance. The maximum total deformation result on the head dish is 1.6 mm indicated in the red location. This study is still acceptable as the conditions are excellent for deformation measurement.

### 3.5 Hydrotest – Equivalent Stress (Von Misses)

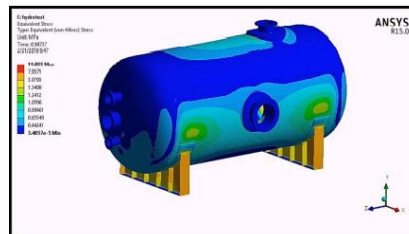
In order to verify the study with a pressure hydrostatic test is to ensure that the pressure vessel can receive a pressure load of 1.5 x MAWP, where the MAWP is 900 psi. Therefore 1350 psi is obtained for a holding time of two hours. At the time of the hydrostatic test, there should be no leakage occurring either in welding or material defects. The medium used for testing is water with a density of 1000 kg/m<sup>3</sup>. Specific weight is the weight ratio to the volume taken into account and is related to the gravitational acceleration in this hydro test analysis. Therefore, the pressure test is proportional to the specific weight of the water. The equivalent von misses stress or ductility resulting from hydro test testing of 11.895 MPa is still far below the yield of the material strength SA 516 Gr. 70 N. The results obtained in this study are excellent. The result of utilization factor is 0.036, where less than 1. Therefore, the analysis result is acceptable.

### 3.6 Hydrotest – Total Deformation

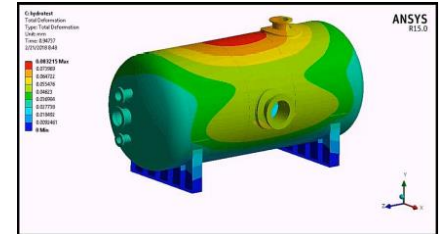
From the study, we generate the maximum deformation is the top of the shell part. It happens because the part is in bending condition or the stiffness value is small compared to others, such as the head and the welded nozzle. After we got an equivalent stress value, it resulted in elastic deformation that occurs in testing the overall pressure vessel part due to the hydrostatic pressure test being equal to 0.08 mm deformation value. The results obtained in this study are good. The overall results obtained are summarized in Table 2.



**Figure 12.** Total deformation of static structural



**Figure 13.** Static structural – Equivalent (Von Misses stress)



**Figure 14.** Hydrotest – Total Deformation

**Table 2.** Summary of Result Analysis

Analysis Method	Utilization Factor				Deformation Value (mm)			
	Head	Nozzle	All Part	Result	Head	Nozzle	All Part	Result
Transient Thermal – Welding Temperature								
Static Structural – Equivalent (Von Misses) Stress	0.548	0.548	-	ACC	-	-	-	-
Static structural – Total deformation	-	-	-	-	1.6	1.6	-	ACC
Hydrotest – Equivalent (Von Misses) Stress	0.036	0.036	-	ACC	-	-	-	-
Hydrotest – Total Deformation	-	-	-	-	-	-	0.08	ACC

### 4. Conclusions

Analysis of the welding heat occurring in the three nozzle clusters confirms whether the condition is safe for operations where this condition is in field actualization is not per ASME VIII Pressure Vessel Code. Emergency conditions are to be selected considering the time of execution at the time of shutdown that requires new modifications to the design of the distance between the nozzles being very close. The best guarantee was to get the results as accurate as possible outside the standard use. This condition is temporary as the pressure vessel must operate. All results have been evaluated as specified in the code and interpreted by the best engineering judgment. Methods requiring a study of stress and deformation occurring in the head section are carefully evaluated so that when a pressure vessel operates at a temperature of 80°F and an operating pressure of 70 – 450 psi must be in a safe state of operation.

### References

- [1] Mazgaj, P., Darnowski, P., Kaszko, A., Hortal, J., Dusic, M., Mendizábal, R. and Pelayo, F., 2022. Demonstration of the E-BEPU methodology for SL-LOCA in a Gen-III PWR reactor. *Reliability Engineering & System Safety*, 226, p.108707.
- [2] Xu, Z., Wang, L. and Jin, M., 2018. Determination of Design Extension Conditions List and Design Optimization of Safety Systems for ADS by Lines of Defenses Method. *IEEE Access*, 6, pp.20788-20794.
- [3] Anggraini, L., 2018. Fatigue Analysis pada Pegas Daun Jenis SST 74 dalam Aplikasi Industri Alat Berat dengan Pemodelan Metode Elemen Hingga. *ROTASI*, 20(2), pp.118-123.
- [4] Moss, D.R., 2004, “*Pressure vessel design manual*,” Elsevier.
- [5] Megyesy, E.F. and Buthod, P., 1997, “*Pressure vessel handbook 10th edition*,” Tulsa Pres. Vssl. Publishing Inc.
- [6] Bozkurt, M., 2022. Towards a unified design-by-analysis solution to pressure vessel nozzle-shell junctions under combined loading.
- [7] Bozkurt, M., Nash, D. and Uzzaman, A., 2020, October. Effect of the internal pressure and external loads on nozzles in cylindrical vessel. In *IOP Conference Series: Materials Science and Engineering* (Vol. 938, No. 1, p. 012007). IOP Publishing.
- [8] Pagitz, M., Bold, J., Baranwal, R.K., Hassan, T. and Agarwal, G., Effect of the Internal Pressure and External Loads on Nozzles in Cylindrical Vessel.
- [9] Kushwah S, Parekh S, Mistry H, Darji J, Gandhi R. Analysis of cylindrical pressure vessels with dissimilar ends and material comparison. *Materials Today: Proceedings*. 2022 Jan 1;51:355-68.
- [10] Arsyad MZ, Widiyanto T. Mechanical Design of Slug Catcher. *Indonesian Journal of Energy and Mineral*. 2021 May 31;1(1):1-2.

- [11] Pathre, S.S., Mathkar, A.M. and Gopalakrishnan, S., 2021, July. Investigation of Nozzle on Knuckle Region of Dished Head. In Pressure Vessels and Piping Conference (Vol. 85314, p. V001T01A067). American Society of Mechanical Engineers.
- [12] Robau-Porrúa A, Pérez-Rodríguez Y, Soris-Rodríguez LM, Pérez-Acosta O, González JE. The effect of diameter, length and elastic modulus of a dental implant on stress and strain levels in peri-implant bone: a 3D finite element analysis. *Bio-medical materials and engineering*. 2020 Jan 1;30(5-6):541-58.
- [13] Utomo, N.K.A. and Anggraini, L., 2019. Finite Element Analysis With Static And Dynamic Conditions Of Spare Wheel Carrier For Oh 1526 Fabricated By Saph 440 Hot Rolled Steel. *Journal of Mechanical Engineering and Mechatronics*, 4(1), pp.34-44.
- [14] Anggraini, L. and Gana, I.B.M., 2020. ANALYSIS OF COMPOSITE ELECTRIC VEHICLE BODY. *Journal of Mechanical Engineering and Mechatronics*, 5(1), pp.1-10.