INVOTEK
Jurnal Inovasi Vokasional dan Teknologi
http://invotek.ppj.unp.ac.id/index.php/invotek
Vol. 22, No. 1, Feb 2022
ISSN: 1411 – 3411 (p)
ISSN: 2549 – 9815 (e)

A Finite Element Analysis of Structural Strength of Ferry Ro-Ro's Car Deck

Muhammad Uswah Pawara[1 *], Alamsyah[1], Rodlian Jamal Ikhwani[1], Ade Ray Siahaan[1], Andi Mursyid Nugraha Arifuddin[1]

[1] Program Studi Teknik Perkapalan, Institut Teknologi Kalimantan
Jl. Sukarno Hatta Km.15 Karang Joang, Balikpapan, Indonesia-76127

*Corresponding author: uswah.pawara@lecturer.itk.ac.id Doi : https://doi.org/10.24036/invotek.v22i1.959

Abstract

Ferry Ro-Ro is a type of ship used to carry passengers and vehicles. This ship has several decks such as navigation decks, passenger decks, and car decks. The car deck is subject to a heavy load as it should accommodate vehicle loads. This study aims to conduct a finite element analysis to determine the stress and deformation experienced on the car deck of the Ferry Ro-Ro and determine the safety factor based on BKI (Indonesian Classification Bureau). The method used in this study is the Finite Element Method (FEM) which has been applied in the finite element based-applications namely Ansys Workhbench. The simulations are carried out using the software by varying the thickness of the car deck's plate, the plate is subjected to pressure loads which are car loads accommodated by the car deck. Moreover, the fixed support is applied in the deck during the simulation. Based on the simulation results of variations of 90, 100, and 110% plate thickness. It is obtained that the smallest plate experienced maximum stress and the obtained safety factor of all plate thickness variations was satisfied the BKI Standard.

Keywords : ferry ro-ro, car deck, finite element method, stress, safety factor

1. Introduction

Ship is a transportation mode operating at sea with a very complex system. The ship construction is built with a comprehensive component as the ship operates in the sea which is a dynamic and unpredictable environment. The ship is vulnerable to dynamic loads such as wind, sea wave conditions, and others so that ships should deal with all conditions in the sea. In addition, the ship should withstand static loads such as cargos and machinery.

Therefore, the complex construction of the ship usually refers to and is supervised by a classification society for example BKI (Indonesian Classification Bureau) In Indonesia. In addition to supervising ship maintenance, such as repair, docking, the classification should insure that during shipbuilding the ship construction complies with rules and regulations. In shipbuilding, the ship is designed to withstand external forces as well as satisfy safety regulation regarding cargo and in particular people safety in the ship.

During operation, all ships experience a heavy and repetitive load, it has an impact on the ship's strength as a result of stress and strain working in the ship structure. Therefore, it is necessary to estimate loads experience by the ship so that excessive load on the ship can be avoided. as a consequence, it prevents from lose due to an accident caused a ship's structure damage. This damage has a disadvantage on ship owners and passenger.

Based on previous studies, the finite element method (FEM) has been applied in the ship structure several years ago. For example, Kamel and Liu discussed FEM application to analyze the ship hull structure [1]. For the most recent studies. Iqbal and Zhifan briefly discussed FEM for complex problems in the ship structure [2], Wang and Wiernicki from the American Bureau of Shipping applied FEM to analyze ice strengthening tankers [3], Tanny et al., used software to perform finite element analyses for structural investigation of container ship's cargo holds [4].

Received date 18/07/2021, Revised date 15/10/2011, Accepted date 01/07/2022

P-ISSN: 1411-3414
E-ISSN: 2549-9815

In the present study, the FEM is used to estimate the working stress and deformation in the car deck. The finite element analysis is performed using a finite element based-application. The simulation is carried out by varying the plate thickness. The investigation of structural strength of car deck has been performed in the previous studies, Tuswan et al., investigated the structural strength of the deck by applying sandwich material on the deck [5], Zaibidi et al., performed the strength analysis on the lengthened car deck [6], Mulyatno et al, carried out investigation on the local stress acting on the car deck [7]. In addition, Alamsyah et al., analyzed the structural strength of Landing Craft Tank (LCT) ship to determine the stress in the deck construction by varying plate thickness [8]. Therefore, this study attempts to perform investigation of structural strength of Ferry Ro-Ro's car deck by varying the plate thickness to determine the stress, this study aims to identify a proper plate thickness that should be installed on the car deck based on BKI standard. The usage of proper plate thickness will ensure shipbuilding cost efficiency.

1.1 Ferry Ro-Ro

Ro-Ro ship had been developed before the second world war (WW II) in several regions in Europe. This ship has been used as a mode of land transportation and it cannot be denied that the initial development of the ship imitated the LST (Landing Ship Tank) which was as an amphibious warfare ship carrying equipment in WW II. under certain conditions especially in situations and conditions with minimum port facilities and poor natural conditions. this ship can still berth.

The ferry Ro-Ro is used to transport Trucks, cars, motorbikes, and passengers. a ferry Ro-Ro has the following general characteristics: 1). The width deck is large enough to accommodate vehicles for the purpose of fluent loading and unloading process. 2) Position the vehicle in such a way as to avoid seawater. 3) The ramp door located forward, aft, and side (port or starboard side).

1.2 Stress

Stress is an internal force that acts over a small and infinite area of a section and consists of various magnitudes and directions. The intensity of the force that is perpendicular or normal to the section is normal stress. Stress is the force acting on a cross-sectional area in a unit. Stress is non-relativistic with units N / m2 or Pascal (Pa). The stress on the material will cause a change in shape [9].

In order to determine stress of the car deck, the first step is calculation of loads acting in the deck, the given load is actual load during ship in operation to produce a comprehensive analysis result, the working load in the car deck is determined by Eq. 1[8].

$$Load_{\ Press} = \frac{Load\ Total}{Deck\ Area} \quad \text{..Eq.1}$$

The next step is determining the working stress of the car deck by Eq. 2

$$\sigma = \frac{F}{A} \quad \text{...Eq.2}$$

Where F is a force acting perpendicular to the cross-section, while A is the area. This normal stress can produce Tensile stress which is caused by tensile loads or loads whose direction is perpendicular to the surface area, Compressive stress which is caused by compressive loads or loads. Shear stress which is caused by a force whose direction is parallel to the surface area [10]. The stress that commonly used in 3D construction model is equivalent stress [11].

$$\sigma_v = \sqrt{\sigma^2 + 3\tau^2} \quad \text{..Eq.3}$$

Where σ_v is equivalent stress, σ is normal stress, and τ is shear stress.

1.3 Finite element Method

The Finite Element Method (FEM) is a numerical method for obtaining solutions to differential problems, both ordinary differential equations (Ordinary Differential Equation) and ordinary differential equations (Partial Differential equation). Because differential equations are often used as models for engineering problems, it is important for engineers to understand and be able to The Finite Element

P-ISSN: 1411-3414
E-ISSN: 2549-9815

Method is a numerical method for obtaining solutions to differential problems, both Ordinary Differential Equation and Partial Differential Equation. Because the differential equation is often used as a model for engineering problems, it is important for engineers to understand and be able to apply FEM [12]. Currently FEM is one of the most versatile numerical methods for solving problems in the continuum domain. Initially FEM was developed to solve problems in the field of solid mechanic, but now FEM has penetrated almost all engineering problems such as fluid mechanic, heat transfer, electro magnetism, vibration, modal analysis, and many other engineering problems [13], and mathematical problems [14].

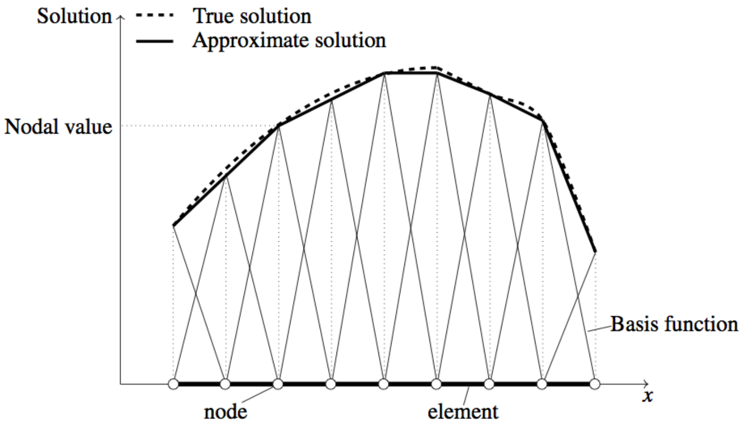


Figure 1 Finite Element Method Principle [15]

The core process of FEM is to divide a complex problem into smaller parts or elements from which simpler solutions can be easily derived. The solution of each element when combined will be a solution to the problem as a whole. Figure 1 explains how FEM works where the solution to a complex problem is approximated by the solution element. To get the elemental solution, FEM uses the interpolation function to approximate the elemental solution. For this example, a simple linear function is used as the interpolation function. After the solution for each element is obtained, by combining the element solutions, the overall solution to the problem can be obtained. By using polynomial functions such as quadratic functions as interpolation functions, more accurate solutions can be obtained [12].

1.4 Safety Factor

The safety factor indicates the ability level of material from external loads which are compressive and tensile loads. The force required for the material to withstand the external load until it breaks is called the ultimate load. By dividing this maximum load by the cross-sectional area, it will obtain the ultimate strength of a material. For the design of structural members, the allowable stress is lower than the maximum strength obtained from the "static" test. The two concepts of allowable stress and ultimate load give the same results for a simple tensile or compressed rod or for more complex structures where failure can be defined by a plastic criterion. In the dissimilar designs, it can be obtained in many cases where the non-resilient nature of the material is taken into account while the failure criterion is excessive plastic deformation [16]. The formulation of the safety factor can be seen in Equation 4.

$$FS=\frac{\sigma\,\text{Ultimate}}{\sigma\,Allowable} \quad \text{......Eq. 4}$$

The allowable stress value is estimated according to the BKI rules Vol.II Section 5 C.1.1 the allowable stress for longitudinal bending stress [17].

$$\sigma p = cs\ .\ \sigma po \quad \text{......Eq. 5}$$

Where σp is allowable longitudinal bending stress, Cs equal to 1, σpo is 18.5 $\sqrt{L/k}$, L is Length of the ship model and, k is yield strength

P-ISSN: 1411-3414
E-ISSN: 2549-9815

2. Methods

In the present study. Numerical simulation software with Finite Element Method (Ansys Workhbench, Student License Study) is used. The type of data used in this study is primary data used in the modeling process of the car deck construction of the ship. The main dimension of the ship can be seen in Table 1. As for support and load on the model, two fixed supports are given on the front and rear sides of the ship model, and the given load is 21 tons which are seven family cars (MPV type) [18].

2.1. Ship Data

The object of research is the Ro-Ro Ferry ship, the ship's car deck will be analyzed to investigate the maximum stress and construction strength. The main dimensions of the ship are shown in Table 1.

Tabel 1 Main dimention of the ship

Particular	Value	Unit
LOA	68.00	m
LPP	62.68	m
B	15.00	m
H	03.50	m
T	02.40	m
Vs	12	Knot

2.2. Ship Design Drawing

The midship section of the ship can be seen in Figure 2 and the car deck has been signed by the red line. Moreover, the detail construction the car deck (main frame and web frame) can be seen in Figure 3. The general arrangement of the car deck and profile view of the ship are shown in Figures 4 and 5 respectively, and the dimensions the car deck are shown in Table 2.

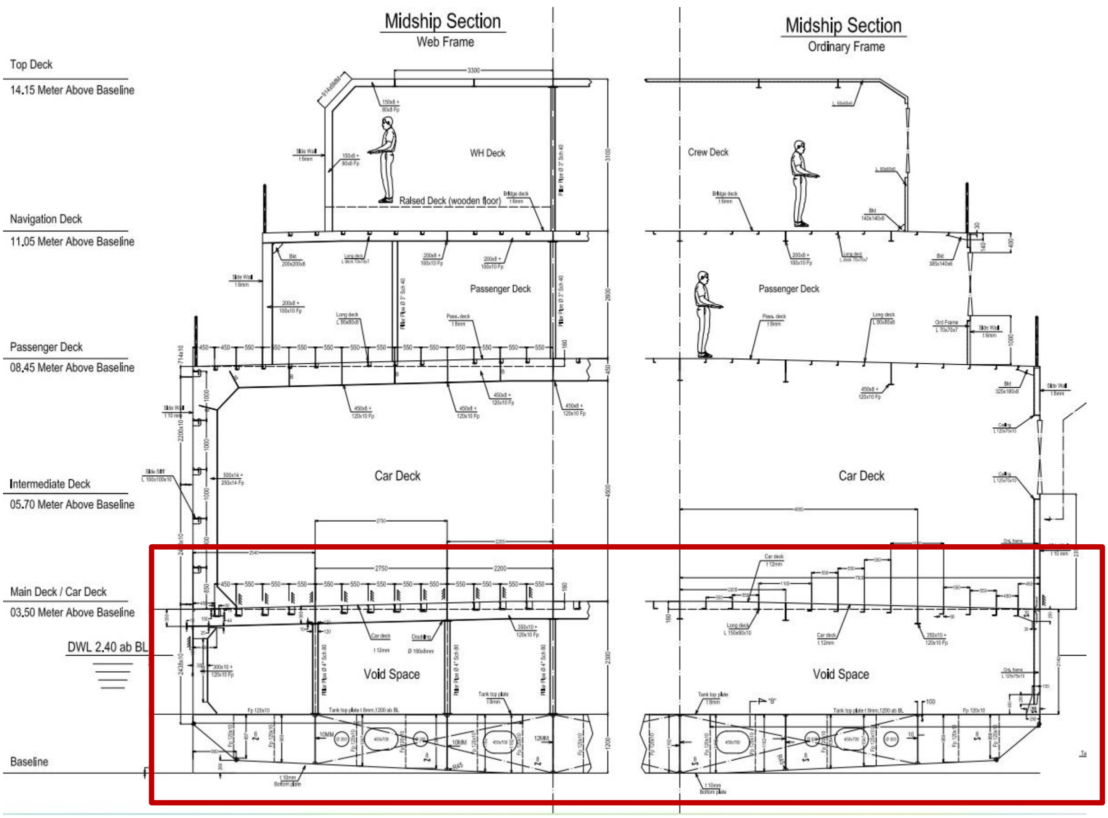
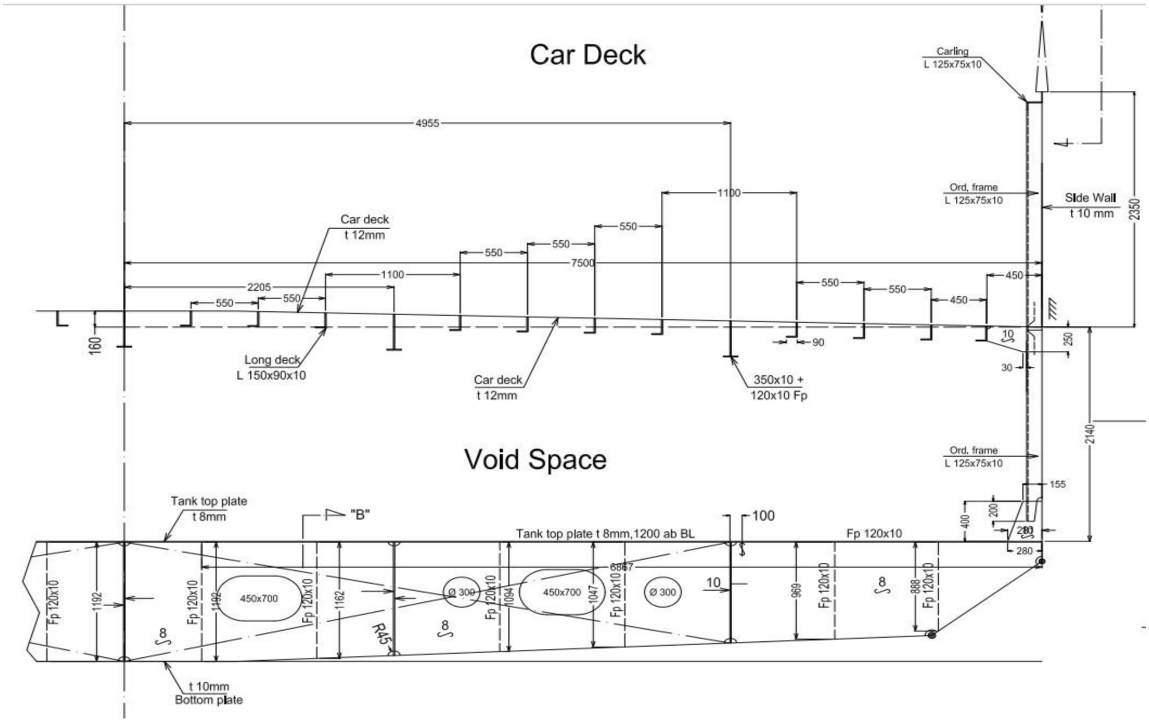


Figure 2 Midship Section

P-ISSN: 1411-3414
E-ISSN: 2549-9815

Main Frame



Web Frame

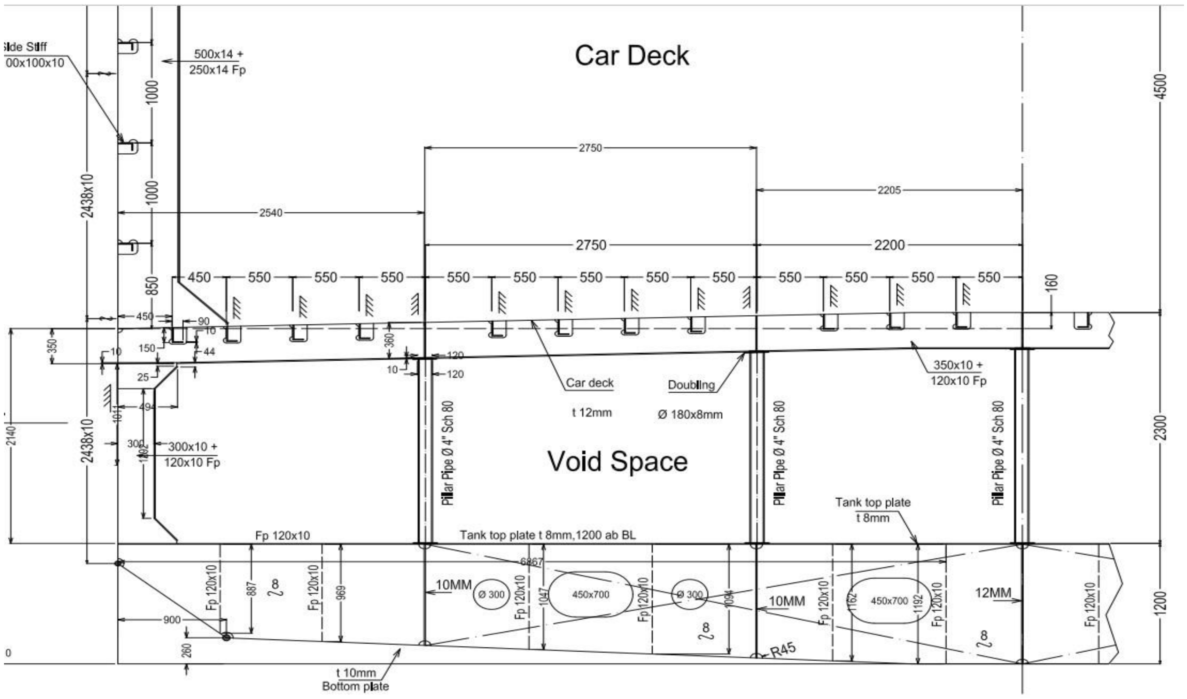


Figure 3 Detail construction of the car deck (main and web frames)

Table 2 The dimension of the car deck

Particular	Value	Unit
Depth (H)	1,1	m
Beam (B)	15	m
Thickness	8	mm

P-ISSN: 1411-3414
E-ISSN: 2549-9815

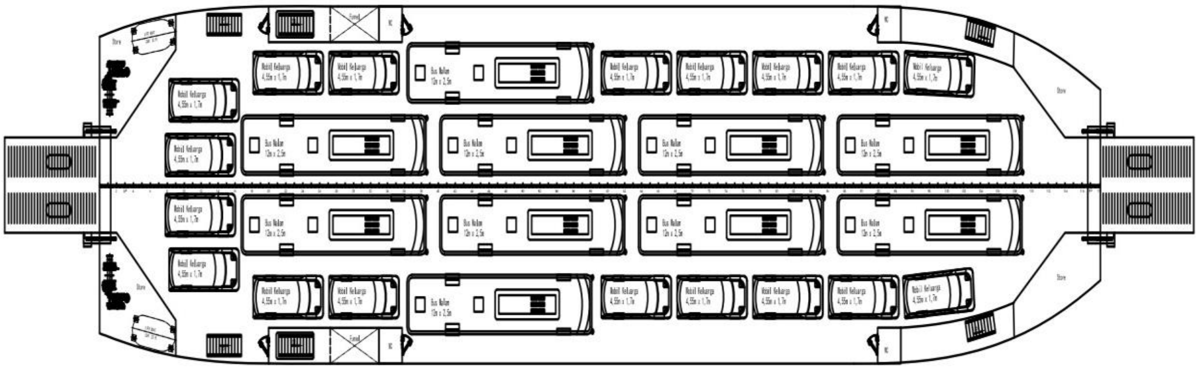


Figure 4 General arrangement of the car deck

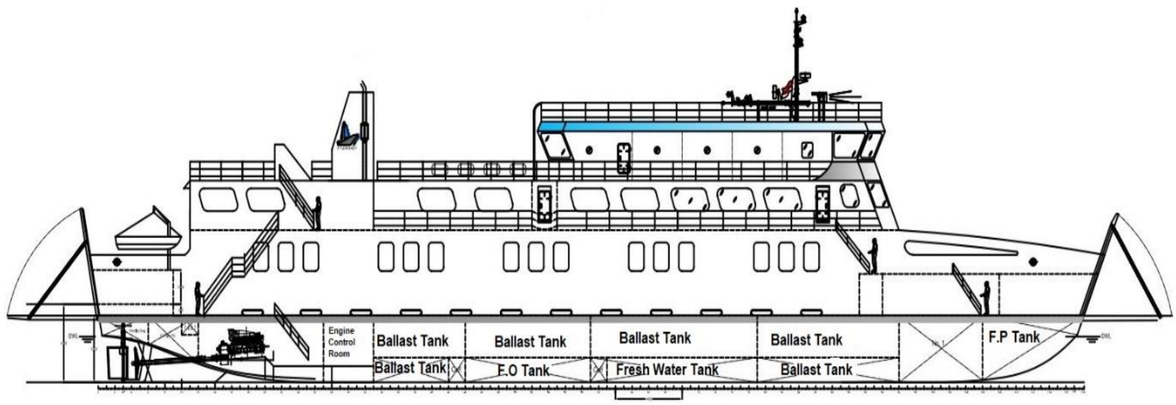


Figure 5 Profile view of the ship

2.3. Material Data

The material for the ship's structure is ASTM A514 Steel with specification in Table 3.

Table 3 Material Data

Component	Value	Unit
Elastic Modulus	210 x 103	N/mm^2
	30,3 x 103	ksi
Yield Strength	690	N/mm^2
	100	ksi
UltimateTensile Strength	938	N/mm^2
	136	ksi
Stain at rupture (ε max)	63%	

2.4. Flowchart

This study is performed based on the schematic methodology which is described by a flowchart. The flowchart reveals the step by step of the research from the beginning until the research completion. The flowchart can be seen in Figure 6.

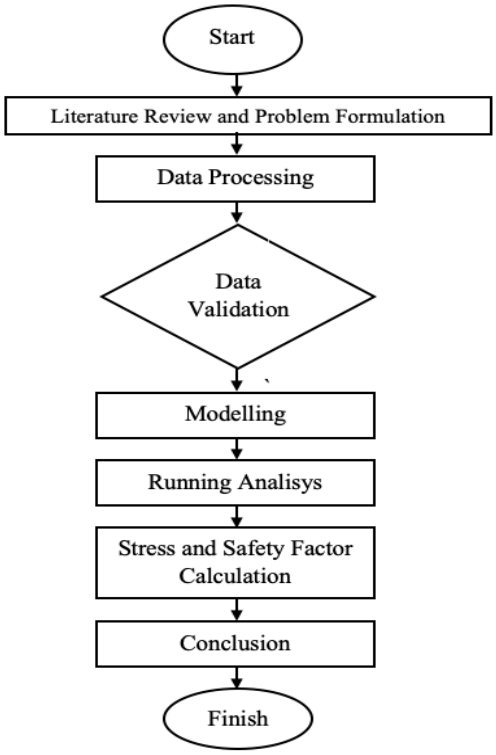


Figure 6 Flowchart

3. Result and Discussion

Validation is a process to find out whether the results of the analysis between manual calculations based on existing theories are in accordance with the calculations carried out on finite element-based software. Validation can be used as a reference whether the results of the analysis that have been carried out are close to true or false, validation is done by comparing the results of calculations in the software and manually. The validation data used can be seen in Table 4 The and simulation results using the application based on the finite element method can be seen in Figure 7.

In the simulation results, the maximum stress that occurs is 1.8049 N / mm2. And the results of manual calculations are obtained from the calculation formula as follows:

$$M\,(max) = \frac{p\,L^2}{8\,Z} \qquad (4)$$

Where: p = Loads (*N/m*)
L = Leangth of Beam (*mm*)
Z = Section Modulus (m^3)

Table 4. Validation of Data

Component	Value	Unit
Length of Beam (L)	5000	mm
Heigh (H)	5	mm
Width (W)	50	mm
Line Pressure Load on Beam (P)	5	N/mm
Youngs Modulus (E)	210000	N/mm^2
Distance from Neutral Axis to Extreme Fibers (C)	30	mm
Moment of Inertia (I)	1080000	mm^4
Section Modulus (Z)	36000	mm^3

P-ISSN: 1411-3414
E-ISSN: 2549-9815

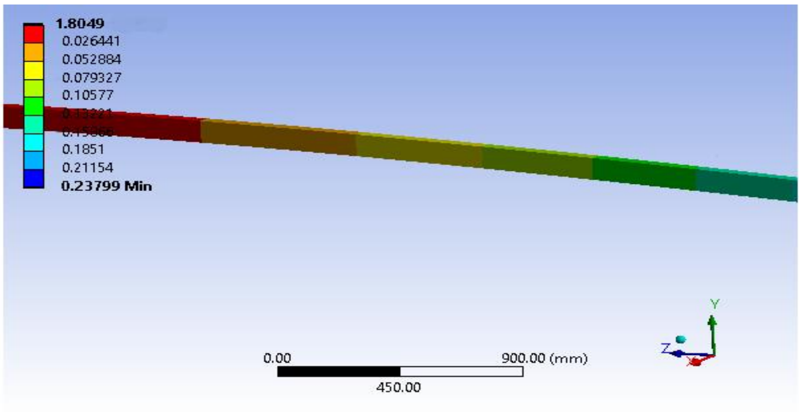


Figure 7 Simulation result

In this modeling, it is used x, y and z axes or commonly called 3D models. Where the axis x represents the width of the model design, the axis z represents the length of the model, and y axis represents thickness or height of the model. As can be seen in Figure 7, the maximum stress is 1.8049 MPa, while the manual calculation is 1.904 MPa. It is clear that the application result is very accurate.

Before analyzing the stress at the car deck in the midship, it is modeled according to the main dimension of the ship. The following are the stages of structural modeling in the software,

1. Run the application.
2. Start a Worksheet.
3. Draw the midship
4. Determine plate thickness
5. Determine the length of
6. Meshing and running

In this study, The Frame 55 to 65 which is 5500 mm is analyzed. Once all dimensions of the model determined. The midship model will be produced as shown in Figure 8. The dimension of the ship model in the modeling stage is a length of 15 m and a beam of 5.5 m

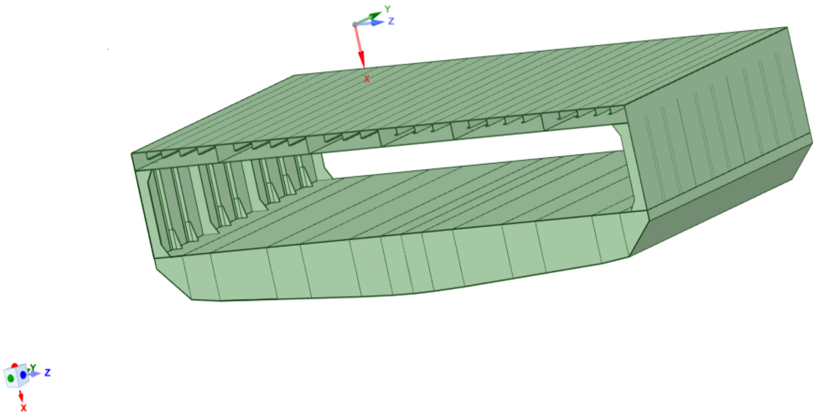


Figure 8 Midship model

The meshing model is the division of the model structure into small elements. The meshing size is given by changing the order element to be linear. Then the element size is determined. In the present study, the element size of 100 mm is given in the messing stage. The meshing result can be seen in Figure 9.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

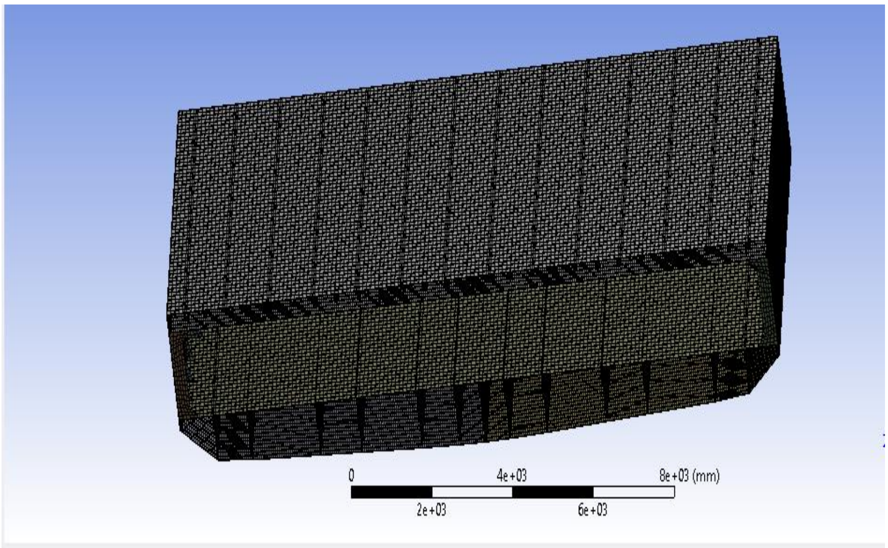


Figure 9 Meshing result

The next stage is to provide support and load on the model. In this modeling, 2 fixed supports are given on the front and rear sides of the ship model. Then Determining the load, the given load is 21 tons (7 family cars) which can be seen in Figure 10. After that, maximum stress and deformation are determined. Once all input in the model is finished, then running the model.

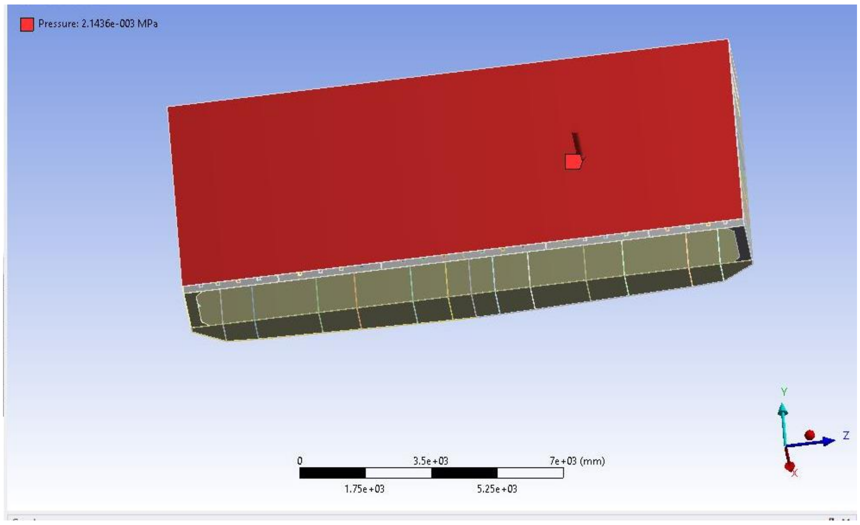


Figure 10 Load Pressure

After running the model structure, the analysis results of the finite element-based application can be seen as follows,

- Plate Thickness of 90%

Stress and deformation can be seen in figures 11, 12, and 13. Based on the Figures with a plate thickness of 10.8 mm and a uniform load of 0.00253527273, the deformation is in the Y-axis with the maximum stress of 21,232 N / mm2 at node 17442. It can be concluded that the greatest stress on each plate thickness variation is in this case. However, the result shows that the stress value is still below the allowable stress value of BKI. This means that the car deck is still safe because the class rules applying a high standard in selecting the construction dimension, although plate thickness has been decreased.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

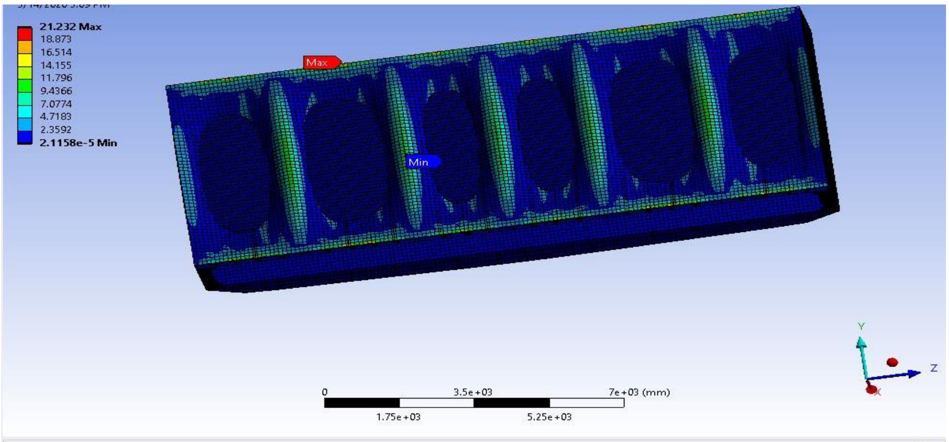


Figure 11 Working Stress of The Thickness of 90%

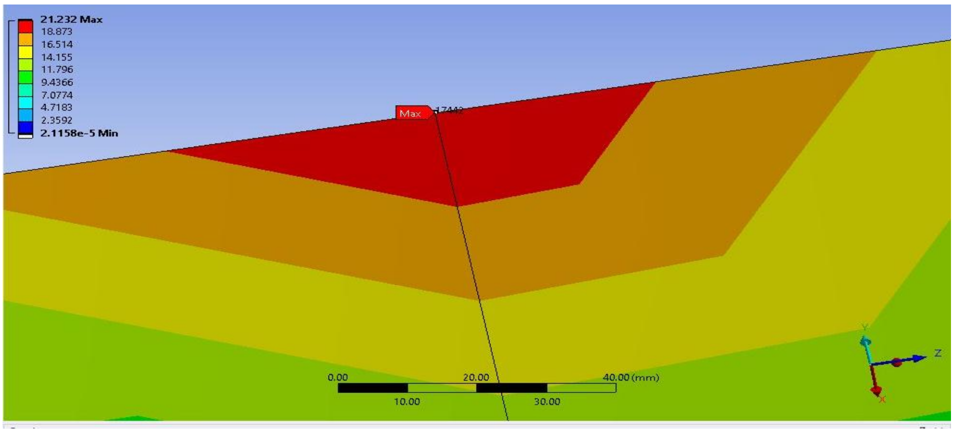


Figure 12 Maximum stress node of the thickness of 90%

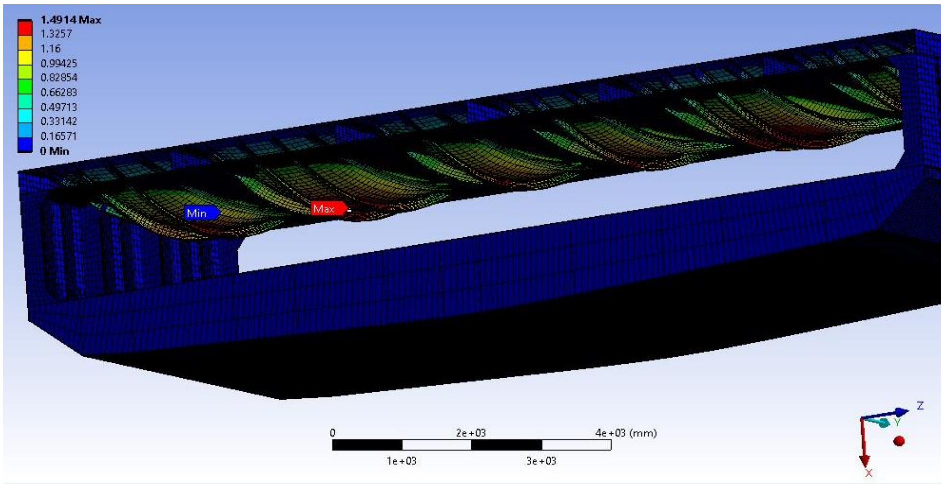


Figure 13 Deformation of the thickness of 90%

- Plate Thickness of 100%

Stress and deformation can be seen in Figures 14, 15, and 16. Based on the figures, with a thickness of 12 mm and a uniform load of 0.00253527273 Mpa. In this condition, it is similar to the previous case. It can be seen that stress number increases in line with the thickness reduction of the deck plate. In this case, the deformation is in the Y-axis with a maximum stress of 21.018 N / mm^2 at node 17442.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

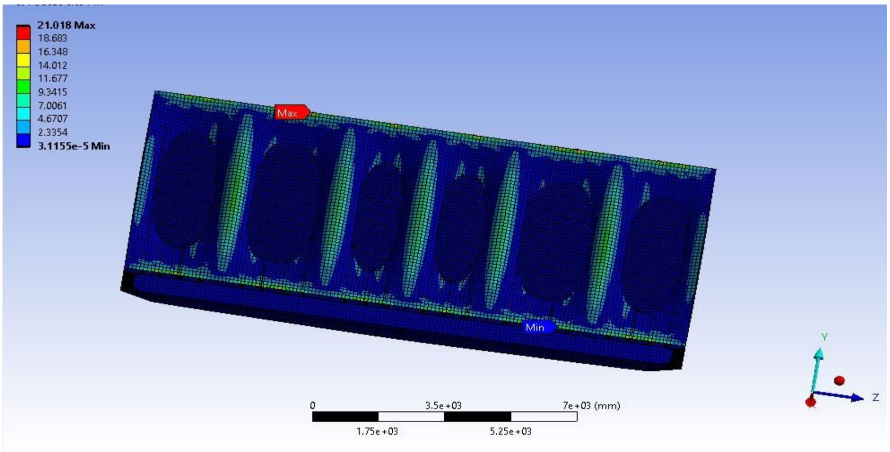


Figure 14 Working stress of the thickness of 100%

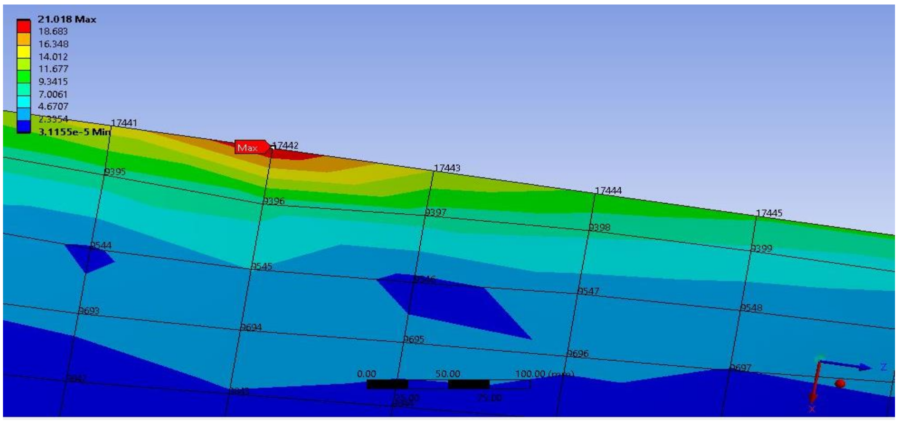


Figure 15 Maximum stress node of the thickness of 100%

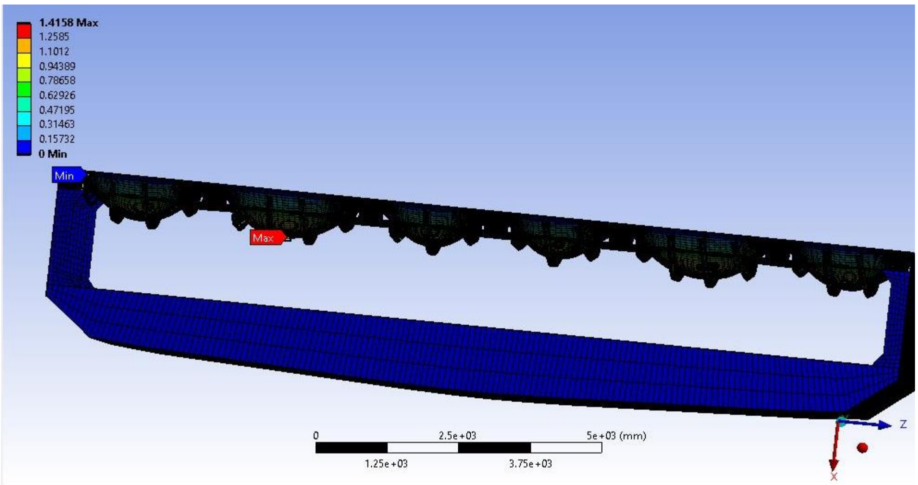


Figure 16 Deformation of the thickness of 100%

• Plate thickness of 110%

The stress and deformation can be seen in figures 17, 18, and 19. Based on the Figures, with a plate thickness of 13.2 mm and a uniform load of 0.00253527273, the deformation experienced on the plate deck is in the Y-axis. From the figure 18, it can be seen the maximum stress occurring at node 17190 with 20,551 N / mm^2. However, the obtained stresses are still below the allowable stress by the BKI standard, as the construction still has a sufficient cross-sectional modulus value.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

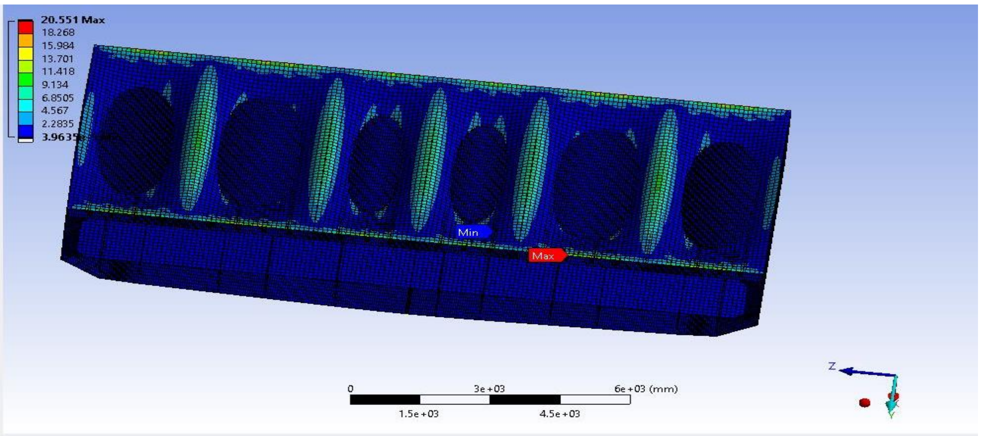


Figure 17. Working Stress in the plat thickness of 110%

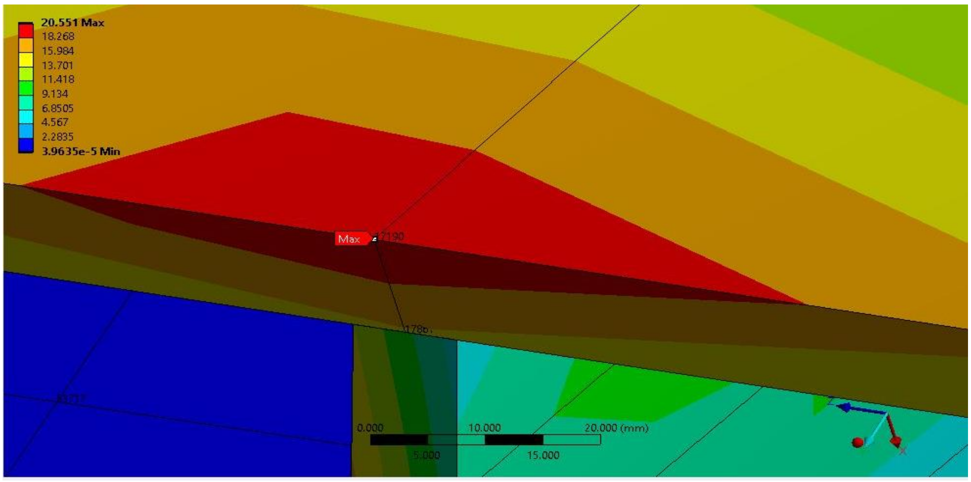


Figure 18. Maximum stress node in the plat thickness of 110%

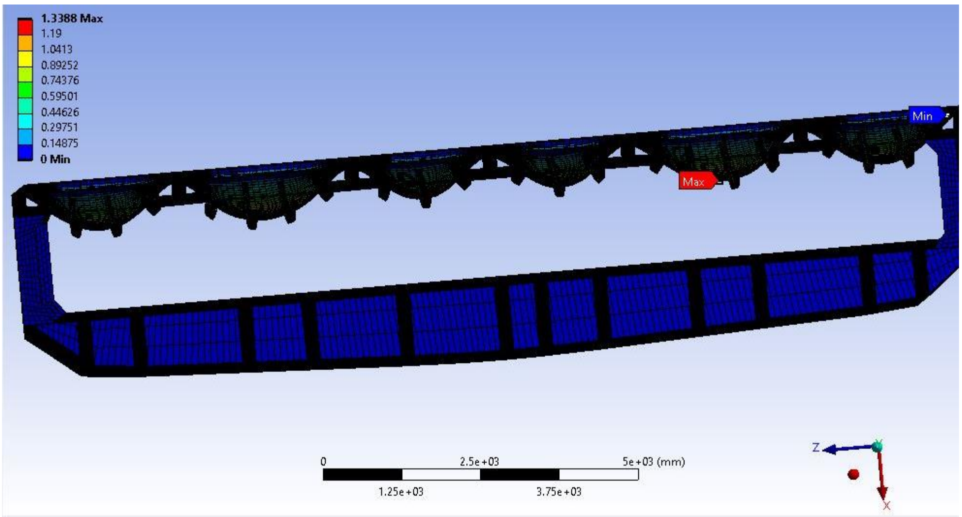


Figure 19. Deformation in the plat thickness of 110%

To estimate the safety factor, the allowable stress should be determined. The allowable stress is calculated according to the BKI Rules Vol. II Section 5 C.1.1 the allowable stress for longitudinal bending stress can be determined using Equation 2. The stress ratio of the plate thickness of 90%, 100%, and 110% can be seen in Table 5.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

Table 5. Estimation of safety factor according to BKI standard

Plate Thickness	Maximum Stress (*N/mm²*)	Allowable Stress (*N/mm²*)	Safety Factor	Condition
90%	21.232	103.840	4.89	Satisfied
100%	21.018	103.840	4.94	Satisfied
110%	20.551	103.840	5.05	Satisfied

4. Conclusion

Based on the finite element analysis of the structure of the car deck of the Ro-Ro Ferry with plate thickness variations and a uniform load of 0.00253527273 Mpa in the Y-axis using the finite element based application, the following conclusions can be drawn. The largest value of maximum stress on the vehicle deck with a plate thickness of 10.8 mm is 21,232 N / mm^2 at node 17442. And the smallest value for maximum stress on the vehicle deck with a plate thickness of 13.2 mm is 20,551 N / mm2 at node 17190. The safety factor values on the plates of 10.8 mm, 12 mm, and 13,2 mm are 4.89, 4.94, and 5.05 respectively. The safety factor values on the plates of 10.8 mm, 12 mm, 13.2 mm are still under the allowable stress and material criteria stress according to the Indonesian Classification Bureau which is 211.04581 N / mm2 and for safety factor values above 1.

References

[1] H. A. Kamel and D. Liu, "Application of the finite element method to Ship Structure," *Comput. Stuctures*, vol. 1, no. 1–2, pp. 103–130, 1971, doi: 10.1016/0045-7949(71)90007-1.

[2] J. Iqbal and Z. Shifan, "Modeling and Simulation of Ship Structures Using Finite Element Method," *Int. J. Ind. Manuf. Eng.*, vol. 12, no. 7, pp. 941–947, 2018.

[3] G. Wang and C. J. Wiernicki, "Using nonlinear finite element method to design ship structures for ice loads," *Trans. - Soc. Nav. Archit. Mar. Eng.*, vol. 113, pp. 103–116, 2006.

[4] T. T. Tanny, N. Akter, and O. M. Amin, "Finite element analysis of container ship's cargo hold using ANSYS and POSEIDON software," *AIP Conf. Proc.*, vol. 1919, no. December 2017, 2017, doi: 10.1063/1.5018530.

[5] T. Tuswan, A. Zubaydi, A. Budipriyanto, and S. H. Sujiatanti, "Comparative Study on Ferry Ro-Ro's Car Deck Structural Strength by Means of Application of Sandwich Materials," no. Senta 2018, pp. 87–96, 2020, doi: 10.5220/0008542800870096.

[6] M. N. Zaibidi, I. P. S. Asmara, P. Nur, and A. Nugroho, "Analisis Kekuatan Konstruksi Car Deck Akibat Penambahan Panjang pada Ro-Ro Passenger Ship," vol. 4, no. 1, 2021.

[7] I. P. Mulyatno and A. Pratama, "Analisa Kekuatan Konstruksi Car Deck Pada Kapal Km. Dharma Ferry 3 Dengan Metode Elemen Hingga," *Kapal J. Ilmu Pengetah. dan Teknol. Kelaut.*, vol. 8, no. 2, pp. 53–61, 2012, doi: 10.12777/kpl.8.2.53-61.

[8] A. Alamsyah, M. D. Arwan, and A. I. Wulandari, "Analisa fatigue life konstruksi geladak pada kapal Landing Craft Tank menggunakan metode elemen hingga," *Turbo J. Progr. Stud. Tek. Mesin*, vol. 10, no. 1, pp. 77–83, 2021, doi: 10.24127/trb.v10i1.1514.

[9] R. Lubis, *Diktat Kuliah Fisika Dasar 1*. JURUSAN TEKNIK INFORMATIKA FAKULTAS TEKNIK & ILMU KOMPUTER UNIKOM, 2008.

[10] Egor P. Popov, *Mechanics of Material Behavior.*, 2nd editio. Prentice-Hall Inc, 1984.

[11] BKI, "Rules for the classification and Construction. Part 1 Seagoing Ship.Volume V Rules for Materials," Jakarta, 2018.

[12] H. Isworo and P. R. Ansyah, "Buku Ajar Metode Elemen Hingga," p. 68, 2018.

[13] N. H. Kim and B. V Sankar, "Introduction to Finite Element Analysis and Design," 2008.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

[14] Y. Susanto, *Dasar Dasar Metode Elemen Hingga*. Yogyakarta: ANDI, 2004.

[15] "4.1.2 Principles of Finite Element Method." https://www.iue.tuwien.ac.at/phd/rovitto/node63.html (accessed Oct. 15, 2021).

[16] Wahyu Dwi Yunanto, I. P. Mulyatno, and A. Trimulyono, "ANALISA KEKUATAN KONSTRUKSI CAR DECK PADA KAPAL 'KAPAL ROPAX 5000GT'DENGAN METODE ELEMEN HINGGA," *J. Tek. Perkapalan*, vol. 2, no. 3, 2014.

[17] Biro Klasifikasi Indonesia, "Rules For Hull Volume II," 2018.

[18] A. R. Siahaan, "ANALISIS KEKUATAN KAPAL FERRY RO-RO MENGGUNAKAN METODE ELEMEN HINGGA," Institut Teknologi Kalimantan, 2020.