Wave: Jurnal Ilmiah Teknologi Maritim
(Journal of Maritime Technology)

Vol. 17, No. 1, July 2023: page: 11-18
p - ISSN: 1978-886X
e - ISSN: 2614-641X

Stress Analysis of 91.5 Metre Coal Carrier Pontoon with Variations of Frame Distance

Khairunnisa[1], Priyambodo Nur Ardi Nugroho[1], and Kharis Abdullah[1,*]

[1]Department of Shipbuilding Engineering, Politeknik Perkapalan Negeri Surabaya, Surabaya, 60111, Indonesia

Article Info

Article history:

Received January 30, 2023
Revised March 22, 2023
Accepted May 10, 2023

Keywords:

Stress Analysis
Allowable Stress
Coal Carrier
Sagging
Hogging

ABSTRACT

The scarcity of materials due to the impact of the pandemic has caused the price of shipbuilding materials is increased and the production costs to become more expensive. So, optimization steps need to be taken to reduce production costs but still ensure the quality of the coal carrier pontoons. One of the optimization steps that can be done is by changing the frame distance. In this study, two variations of the frame distance, 600 and 650 mm, will be carried out to obtain the optimal value from the initial design of 610 mm. In the numerical calculation, each construction model will be analyzed for the magnitude of the stress using finite element method software in still water conditions, sagging, and hogging. From the calculation results, the profile size of the frame distance of 600 mm is smaller than the frame distance of 650 mm; this happens because the modulus value is smaller. In the stress analysis, the highest allowable stress value was obtained from the sagging condition at a frame distance of 610 mm with a value of 84.87 MPa.

Corresponding Author:

Kharis Abdullah
Department of Shipbuilding Engineering, Politeknik Perkapalan Negeri Surabaya, Surabaya, 60111, Indonesia
Email: kharis.abdullah@ppns.ac.id

INTRODUCTION

In supporting world trade, one of the reliable routes is sea lanes, so the presence of ships is urgently needed. To help this, the ship's construction must be robust enough to accommodate the cargo to its destination safely. However, ship construction must also be light to maximize its economic value. Therefore, ship construction efficiency is needed to accommodate this goal.

In the context of construction efficiency, one of the things that can be done is to make changes to the spacing of the ship's hulls. Changes in the frame spacing are allowed to be carried out provided that they do not exceed the maximum limit of the frame spacing according to Biro Klasifikasi Indonesia (BKI) and also that the tension of the ship is still below the permissible stress. In this study, the reduction and addition of frame spacing was carried out from 610 mm (existing construction) to 600 and 650 mm.

In this study, the pontoon is a coal pontoon with no crew and engines inside, so to move places, it needs to be towed by tugboats. Pontoons or commonly called barges are cargo ships with bulk cargo types but are not equipped with a propulsion system (Utomo, 2018).

The ship numerical modelling was carried out by modelling the amidships using the finite element method. It was taking this centre section because the middle area of the ship has the maximum stress value of all parts (Adnyani et al., 2019). In carrying out numerical modelling, conditioning is required, which is close to the reality in the field. In this study, we input sea conditions during sagging, hogging, and still water and include the hydrostatic pressure on the ship. From the numerical modelling results, it can be compared whether the changes in the spacing of the frame made still meet the allowable stress required by BKI.

METHODOLOGY

The study followed a methodology presented in Figure 1. Coal Carrier Pontoon in this study was selected with existing construction having a distance between frames of 610 mm. Then the construction calculation is carried out per the rules, where the frame's distance is 600 and 650 mm. Modelling is carried out by using finite element method software. Modelling results with software were compared with the minimum stress allowed by the rules.

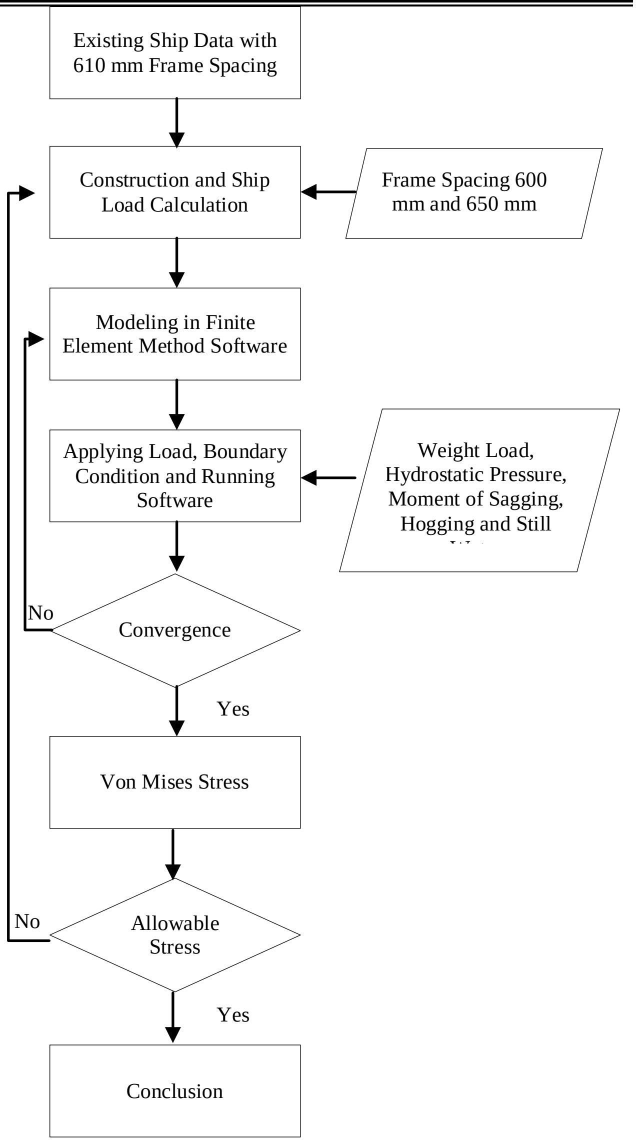

Figure 1. The methodology of research

Construction

Coal carrier pontoon construction is calculated by using the rules of the Indonesian Classification Bureau, called BKI, where the new construction uses a frame spacing of 600 and 650 mm (BKI, 2017).

Finite Element Method

The finite element method is a numerical method widely used to solve various engineering and mathematical problems (Logan, 2007). This method can solve various problems such as structural strength, fluid, electromagnetic, etc. In the method, a structure or construction will be

Vol. 17, No. 1, July 2023: page: 11-18
p - ISSN: 1978-886X
e - ISSN: 2614-641X

divided into small elements and transform these elements into a mathematical model.

At this time, much software was developed to facilitate calculations with the finite element method. In general, the Finite Element Analysis (FEA) modelling software has six stages, including selecting the type of analysis, selecting the element type, determining the material properties, modelling, determining the loads and boundary conditions, and interpreting the results (Abdullah et al., 2018).

Ship construction is a structure that has high complexity, so many numerical or computational methods are used to determine and evaluate the strength of the structure (Abdullah & Santosa, 2019). The complex ship construction causes complexity in modelling, so simplification is needed by selecting several constructions to be analyzed (Abdullah et al., 2023).

Convergence

In using the finite element method software, the more elements used, the more accurate the analysis results, but this is also in line with the longer time needed to process the analysis. So it is recommended to determine the number of elements whose accuracy value is acceptable. Convergence determines how a model produces an analysis that does not change the number of elements so that it can be said how many elements are needed in a model (Abdullah et al., 2023).

According to Cook (1995), a problem model is considered convergent if the difference in the convergence test results is approximately 5%.

Load Calculation

Loads that occur on ships are caused by static and dynamic loads, which can cause changes in the shape of the ship and can cause stress on the ship (Wulandari et al., 2021). The cargo load on this ship is placed on the ship deck. In this modelling is assumed to be a uniform load. The calculation of the ship's cargo load starts with calculating the ship's total cargo and the area of the ship's cargo area; from these results, the load is obtained per square meter. From the total load, it is identified how much load is borne by each model based on the area of cargo area in each model.

Moment Calculation

A bending moment occurs when a force is applied to a structure from a specific reference point, causing a bending effect on the structure. If the structure is not suspended, it will cause the structure to rotate at a certain point (Wulandari et al., 2021). This dynamic load calculation is carried out when the ship is in sagging and hogging conditions. The bending moment loads in this study were obtained from the longitudinal strength analysis performed in the stability software.

This modelling is carried out in frame number 20 to frame number 32; modelling is carried out in the midship area, where this area is most dangerous when the water conditions are wavy (Wulandari, 2022), as shown in Figure 2, where this research will be modelled on the construction between frames 20 to 32.

Boundary Condition

Boundary conditions or supports are the place or sections where a reaction takes. Determining the boundary conditions is necessary so that the modelling is by the existing conditions (Abdullah et al., 2023). The location

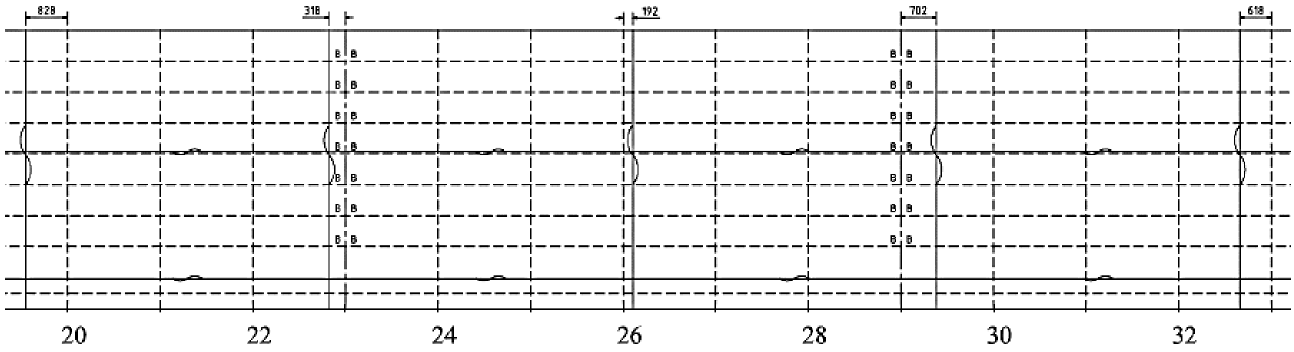

Figure 2. Construction profile frame 20-32

Location	Translation			Rotation		
	δ_x	δ_y	δ_z	θ_x	θ_y	θ_z
Aft End						
Independent point	-	Fix	Fix	$M_{T\text{-}end}$ (4)	-	-
Cross section	-	Rigid link	Rigid link	Rigid link	-	-
	End beam, see [3.2.2]					
Fore End						
Independent point	-	Fix	Fix	Fix	-	-
Intersection of centreline and inner bottom (3)	Fix	-	-	-	-	-
Cross section	-	Rigid link	Rigid link	Rigid link	-	-
	End beam, see [3.2.2]					

Note 1: [-] means no constraint applied (free).
Note 2: See Figure 17.
Note 3: Fixation point may be applied on other continuous structures such as outer bottom at centreline. If exists, the fixation point can be applied at any location of longitudinal bulkhead at centreline, except independent point location.
Note 4: hull girder torsional moment adjustment in kNm, as defined in [6.4.4].

Figure 3. Boundary conditions according to DNVGL-CG-0127

of the structure boundaries to be analyzed is placed in the fixed geometry area (Trihantoro et al., 2022). Determining these boundary conditions refers to DNV (DNVGL, 2015), as seen in Figure 3.

RESULTS AND DISCUSSION

In this study, the pontoon or coal barge has dimensions of a Length over All (LoA) of 91.5 m, a Breadth (B) of 24.4 m, and a height (H) of 5.5 m, as shown in Table 1.

Table 1. Geometric properties of the coal barge

Principal Dimension	Values (m)
Length over All (LoA)	91.5
Breadth (B)	24.4
Height (H)	5.5

In Table 2, it can be seen the detail of the existing construction of a pontoon or coal barge. The bottom, deck and side longitudinal have the same profile size, L120x120x11 mm. Table 3 and Table 4 show the construction of barges with new designs, where construction calculations

Table 2. Existing construction

Construction Section	Profile (mm)
Bottom Longitudinal	L120x120x11
Deck Longitudinal	L120x120x11
Side Longitudinal	L120x120x11
Web Frame	T450x75x10
Strut	L130x130x12
Bulkhead Stiffener	L130x130x9
Side Stringer	L550x100x10

refer to the rules of the BKI (2017). According to the calculations that have been carried out, the size of the design with a 600 mm frame spacing has a smaller profile size than the design with a 650 mm frame spacing which results in a more extensive profile size, this is due to the unsupported span of the 600 mm frame spacing design is smaller than the 650 mm frame spacing design, so the profile modulus of 600 mm frame spacing is smaller than the 650 mm modulus profile. Details of the load calculation can be seen in Table 5.

Table 3. Construction with 650 mm frame spacing

Construction Section	Profile (mm)
Bottom Longitudinal	L150x90x12
Deck Longitudinal	L150x90x9
Side Longitudinal	L150x90x12
Web Frame	T350 x13x300x24
Strut	L200x200x20
Bulkhead Stiffener	L150x150x15
Side Stringer	L200x200x15

Table 4. Construction with 600 mm frame spacing

Construction Section	Profile (mm)
Bottom Longitudinal	L125x75x10
Deck Longitudinal	L100x75x10
Side Longitudinal	L125x75x10
Web Frame	T350 x12x175x19
Strut	L200x200x15
Bulkhead Stiffener	L150x150x12
Side Stringer	L200x200x15

As shown in Table 6 to Table 8, the moments that occur in the existing construction and at the frame spacing of 600 and 650 mm.

Table 5. Load on the barge

Model	Pressure (N/m^2)
Existing	12,251.54
Frame Spacing 650 mm	13,041.52
Frame Spacing 600 mm	12,086.92

Table 6. Moments on existing construction

Section	Momen (N.m)		
	Sagging	Hogging	Still Water
Frame 20	-3.89E+08	4.57E+07	-1.86E+08
Frame 32	-3.10E+08	5.09E+07	-1.46E+08

Table 7. The moment at 600 mm frame spacing

Section	Momen (N.m)		
	Sagging	Hogging	Still Water
Frame 20	-3.80E+08	4.94E+07	-1.79E+08
Frame 32	-3.24E+08	5.13E+07	-1.53E+08

Table 8. The moment at 650 mm frame spacing

Section	Momen (N.m)		
	Sagging	Hogging	Still Water
Frame 20	-4.24E+08	4.15E+07	-1.99E+08
Frame 32	-2.57E+08	5.12E+07	-1.15E+08

Simulation in Finite Element Software

The finite element software simulation consists of several stages: identifying the materials used, conducting meshing, giving loads and boundary conditions, and selecting solutions. After the simulation, each model's stress results will be obtained. One of the results can be seen in Figure 4.

In selecting the element type, the element type is set automatically by following the software program with the typical Program Controlled; the element type of the model is Linear Elements. In the convergence test of this research, the convergence test was carried out on a model of pontoon model with a frame spacing of 650 mm, as shown in Table 9. Based on the program control, the number of nodes increases as the mesh size is smaller, but the number of elements is nearly stagnant between mesh sizes.

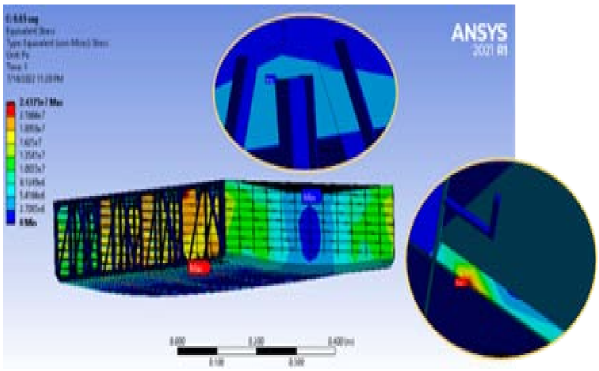

Figure 4. Stress simulation results in finite element software

Table 9. Size mesh and node number

Size Mesh (mm)	Node Number	Element Number	Von Mises Stress (MPa)
1000	102595	31981	22.10
600	102595	31981	22.08
500	102632	31997	22.10
400	102630	31995	22.07
300	103733	32403	23.40

Then a convergence test is carried out to ensure the stress value can be used as a reference. This convergence process is principally carried out by changing the size of the elements in the model and comparing the analysis results with the sizes of other elements; from this, it will be seen whether the analysis results have a stable value. If the analysis value shows a stable value or the curve has formed a straight line, then the model has converged. The result of this convergent stress will be analyzed for the allowable stress. The results of the convergence of one of the models can be seen in Figure 5. Based on the convergence test results, this study uses a mesh size of 500 mm.

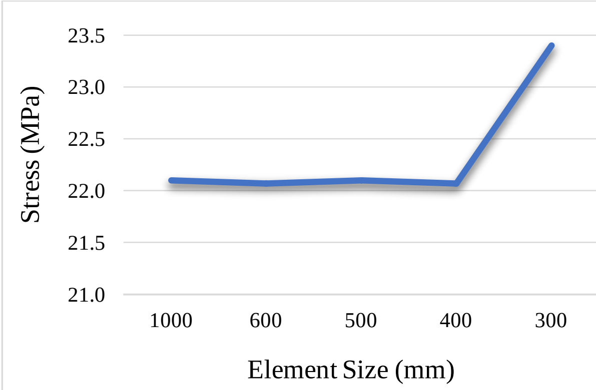

Figure 5. Convergence test results

Stress Analysis

The following is the calculation of the permissible stress allowed by BKI (2017). For strength assessment using finite element analysis from BKI, the Von Mises stress is less than equal to the nominal upper yield of the material, as shown in Equation (1).

$$\sigma_v \leq R_{eH} \quad (1)$$

R_{eH} is Nominal upper yield point of the material used (N/mm^2).

The coal carrier pontoon structure uses regular strength hull structural steel in this construction. It is a hull structural steel with a minimum nominal upper yield point ReH of 235 N/mm^2. So, the Von Mises stress is less than equal to 235 N/mm^2, as shown in Equation (2).

$$\sigma_v \leq 235\ N/mm^2 \quad (2)$$

The results of these calculations become a reference for assessing the stress of each model obtained in Finite Element Software. So from this, the allowable stress analysis is obtained, which can be seen in Table 10.

Table 10. Stress results in each construction

Condition	Von Mises Stress Result (MPa)		
	Existing Construction	Frame Spacing 600 mm	Frame Spacing 650 mm
Sagging	84.80	48.13	24.60
Hogging	67.40	53.60	22.10
Still Water	59.39	49.36	24.52

Table 10 shows that all models have passed the BKI allowable stress and have a stress below 235 N/mm^2. The table shows that the 650 mm frame spacing model achieves the model with minor stress, and the highest stress is obtained by the 610 mm frame spacing or the existing construction.

CONCLUSION

The conclusion that can be obtained from the research is that the change in profile size at a spacing of 600 mm frame tends to be smaller than the profile of 650 mm frame spacing. This is due to the unsupported span, so the modulus profile value at 600 mm frame spacing is smaller at the same profile position. Conditions around the ship during sagging, hogging and still water affect the amount of stress experienced by the ship, as seen from changes in the allowable stress values that occur in each condition. The highest allowable stress value is obtained from the Sagging condition at 610 mm frame spacing with a value of 84.8 MPa.

REFERENCE

Abdullah, K., & Santosa, B., 2019. Analisis Kekuatan Konstruksi Graving Dock Gate Menggunakan Metode Elemen Hingga. *Kapal: Jurnal Ilmu Pengetahuan dan Teknologi Kelautan*, 16(2), pp. 41-48.

Abdullah, K., Sumardiono, S., & Soeroso, H., 2023. Strength Analysis of the Deck Crane Barge Using the Finite Element Method. *Proceedings of the 1st International Conference on Sustainable Engineering Development and Technological Innovation, ICSEDTI 2022*, Tanjungpinang, October 11-13, pp. 1-12.

Abdullah, K., Zubaydi, A., & Budipriyanto, A., 2018. Aplication of Sandwich Plate System with Core from Waste Clam Shell in Ship Deck. *Jurnal Wave*, 12(2), pp. 53-60.

Adnyani, L. P., Aisyah, N., Sulistijono, S., Harahap, R. G., Dianiswara, A., & Nurmawati, N., 2019. Analisa Kekuatan Deck Barge Nania 10070 DWT dengan Dua Metode Pembebanan. *Kapal: Jurnal Ilmu Pengetahuan Dan Teknologi Kelautan*, 16(2), pp. 49-55.

Biro Klasifikasi Indonesia (BKI), 2017. *Rules For the Classification and Construction Part 1 Seagoing Ships.* Biro Klasifikasi Indonesia Volume II Rules For Hull, Jakarta.

Cook, R. D., 1995. *Finite Element Modeling For Stress Analysis*. New Jersey: John Wiley & Sons.

DNVGL, 2015. *Finite Element Analysis.* DNV GL Class Guideline DNVGL-CG-0127, Cape Town.

Logan, D. L., 2007. *A First Course in the Finite Element Method* (4th Ed.). Toronto: Thomson.

Trihantoro, A., Mulyatno, I. P., & Amiruddin, W., 2022. Analisa Kekuatan Struktur Deck Crane Kapal Tanker 6500 DWT Menggunakan Metode Elemen Hingga. *Jurnal Teknik Perkapalan*, 10(4), pp. 52-58.

Utomo, E., 2018. Perbandingan Desain Dermaga Terapung pada Sistem Konstruksi Melintang dan Memanjang "Studi Kasus Dermaga Pelabuhan Tideng Pale Kabupaten Tana Tidung." *Borneo Engineering Jurnal Teknik Sipil*, 2(1), pp. 1-13.

Wulandari, A. I., 2022. Longitudinal Strength Analysis on Deck of Container Ship 409 TEU with Finite Element Method. *Wave: Jurnal Ilmiah Teknologi Maritim*, 16(1), pp. 17-22.

Wulandari, A. I., Alamsyah, & Agusty, C. L., 2021. Stress Stain Analysis on Deck and Bottom Plate of Ferry Ro-Ro Ship with Finite Element Method. *Wave: Jurnal Ilmiah Teknologi Maritim*, 15(1), pp. 45-52.