Wave: Jurnal Ilmiah Teknologi Maritim **Vol. 17, No. 2, December 2023: page: 65-74**
(Journal of Maritime Technology) p - ISSN: 1978-886X
e - ISSN: 2614-641X

Construction Strength Analysis of a 250-Tonne Capacity Deck Crane Barge with Longitudinal Variation

Kharis Abdullah[1,*], Sumardiono[1], M. Lukman Arif[1], Agung Prasetya Utomo[1], and Denny Oktavina Radianto[1]

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

Article Info

Article history:

Received December 14, 2023
Revised January 18, 2024
Accepted January 23, 2024

Keywords:

Deck Barge
Crane Barge
Construction
Finite Element
Strength Analysis

ABSTRACT

A deck crane barge is a barge with a crane system attached to the deck for loading, unloading, and lifting various materials or supporting equipment. In marine building construction, strength assessment must be carried out to ensure that the built design will not fail. The finite element method is one method to perform strength assessment of ship construction faster and simplify calculating. This study assessed the strength of crane barge decks with three variations of longitudinal size with reduced longitudinal size on the web and face plate with the element method. From the calculation results, Model A, or the existing construction, has the smallest Von Mises Stress, while Model C, with reduced web and face plate size to L 80 x 80 x 8 mm, has the largest Von Mises Stress. The reduction in the size of the faceplate and web plate in the longitudinal section reduced the strength of the ship construction with higher stress values. However, all three tested Models had Von Mises Stresses below the maximum permissible stress required by BKI.

Corresponding Author:

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

INTRODUCTION

The marine industry is experiencing rapid development in the development of various types of transport equipment. Various means of transportation were created to facilitate moving goods or people and develop various facilities to support existing industrial processes, such as inspection robots, unmanned surface vehicles, etc. In the construction of marine buildings, a tool or facility is needed to support the construction process; one of the supporting tools is the deck crane barge, a barge with crane facilities. A deck crane barge, or a crane barge, is a specialized barge with a crane system attached to the deck for loading, unloading, and lifting various materials or supporting equipment.

Crane barges are often used when building marine buildings due to their excellent ability, with a tolerance of 50 mm when installing the jacket platform. The type of crane barge can be shear leg cranes or fully rotating cranes. (El-Reedy, 2012). According to their capacity, deck crane types can be divided into two types: Simple cranes and Derrick Barge. Simple cranes are flat-hulled barges with one or more simple lifting cranes installed; with larger crane configurations, this type of deck crane barge can carry up to 2000 tons. This type has a fixed or rotating crane. Derrick Barges can carry huge loads and operate as a fully rotating crane system. This type is used for offshore work as well as for the repair of various marine buildings. On the other hand, the advantage of using a deck crane barge is that it can be used in various water conditions, is flexible in its use, and can also be used to assist the cargo handling process. (Konecranes, 2019). As shown in **Error! Reference source not found.**, the design of the deck crane barge with a rotating crane. In another case, a deck barge or floating crane barge can be used for decommissioning offshore structures (Firdaus et al., 2021).

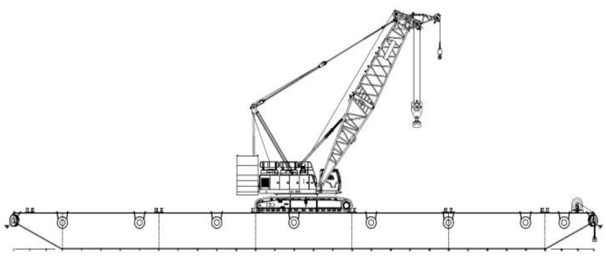

Figure 1. Design of deck crane barge

During the design process of a marine project, especially the construction of marine structures, a mandatory step must be performed, which is the strength check process. This step ensures that the structure and construction have the structural strength that meets the requirements. This process aims to avoid structural damage, which should be avoided during the construction design process. Various structural failure factors can cause vessel damage and affect operations (Soares, 2011). An example of a ship's structural failure is a deck unable to withstand the loads present during ship operations (Wulandari et al., 2022). Marine structures must comply with design requirements to minimize the probability of damage (Vukelić & Vizentin, 2017). The durability of marine building structures affects the structure's durability, where structural durability is the ability of the structure to maintain its mechanical performance over its service life. Structural analysis is necessary under various static and dynamic loading conditions to create a safe and reliable shipbuilding design. Shipbuilding is a rather complex construction system, so the analysis process is generally carried out by selecting several parts of the construction (Ertas et al., 2014).

Experimental and numerical methods can be used to calculate a structure's strength. Nowadays, various methods are developed to facilitate structural or engineering calculations. One of them is a numerical method using the Finite Element Method (FEM), one of the most used methods in structural analysis. FEM has the advantage of its wide adaptability, e.g. its application to problems involving heterogeneous materials, containment of complex geometric structures, and various types of nonlinearities. In addition, the finite element method simplifies the analysis process. Finite element method calculations can be performed by numerical simulation with a computer using software. Examples of calculation software using the finite element method are Abaqus, NASTRAN, SAP 2000, ANSYS, LS-DYNA, and others. Using the capability calculation of FEM software is quite effective and fast (Logan, 2007). This study will discuss the structural strength of deck crane barges using the finite element method software.

METHOD

In this study, the detailed analysis model of the deck crane barge structure is divided into two parts, the model details and FEM.

Model Details

Typically, the materials used in shipbuilding are A36 steel. In this study, deck crane barges use A36 steel, which A36 steel generally has material properties as shown in Table 1, the Young's modulus of A36 steel is 260 GPa, the maximum tensile strength is 400 MPa, and Poisson's ratio coefficient is 0.26 and density 7,850 kg/m^3.

In this study, the deck crane barge has dimensions of a Length between perpendiculars (LBP) of 54 m, Breadth molded (B) of 18 m, a Depth molded (D) of 3 m, and a Draft molded (T) of 2 m, as shown in Table 2.

The midship section of the deck crane barge is shown in Figure 1**,** where the construction uses a longitudinal construction system. The dimensions of the plate and profile sizes in this construction are shown in Table 3. In this study, there are three construction models to be analyzed, where each model has a different longitudinal size. Model A, is a model with existing construction, while models B and C have different longitudinal sizes. As shown in Table 4, where in models B and C there is a reduction in the size of the longitudinal.

Table 1. Material properties of grade A36 steel

Material Properties	Values	Unit
Density	7,850	kg/m^3
Ultimate Tensile Strength	400	MPa
Young Modules	260	GPa
Poisson's ratio	0.26	

Table 2. Geometric properties of the barge

Principal Dimension	**Values**	**Unit**
Length between perpendiculars (LBP)	54	m
Depth molded (D)	3	m
Breadth molded (B)	18	m
Draft molded (T)	2	m

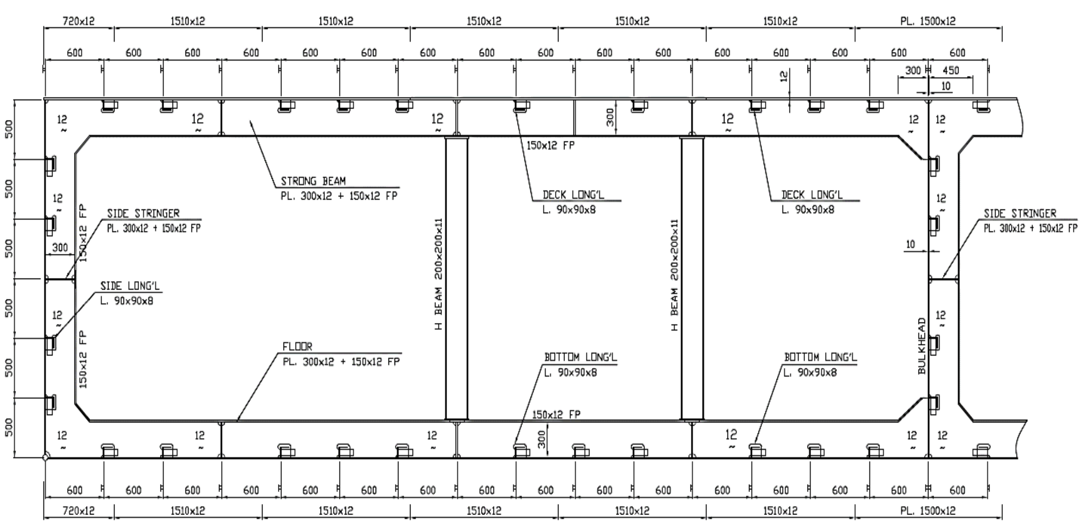

Figure 1. The midship section

Table 3. Plate and profile construction

Part Construction	Plate or Profile (mm)
Side Plate	10
Side Longitudinal	L 90x90x8
Side Stringer	PL. 300x12 + 150x12 FP
Bulkhead Plate	10
Bulkhead Longitudinal	L 90x90x8
Bulkhead Stringer	PL. 300x12 + 150x12 FP
Deck Plate	12
Deck Longitudinal	L 90x90x8
Deck Beam	PL. 300x12 + 150x12 FP
Bottom Plate	12
Bottom Longitudinal	L 90x90x8
Bottom Transversal	PL. 300x12 + 150x12 FP
Pillar	H 200x200x1

Table 4. Model construction

Model	Longitudinal Size (mm)
Model A (existing)	L 90x90x8
Model B	L 85x85x8
Model C	L 80x80x8

Finite Elements

The finite element method is a numerical method widely used in engineering, such as structural strength, fatigue, material damage, and others. Typically, calculations using the finite element method are supported by software. Modeling in finite element method software has several steps. These steps must be performed correctly for the model to run or be calculated with the computer (Logan, 2007), three stages of calculating with finite element software are: pre-processor, solution, and post-processing.

In this study, the loads acting on the structure are hydrostatic loads and crane-weight loads. The load applied to the crane barge is applied in calm water conditions without wave interference. Hydrostatic pressure occurs at a water level of two meters, and the load exerted by crane pressure is to the specifications set by the manufacturer or the crane catalog. As shown in Figure 2, the crane barge load includes the crane's weight and the hydrostatic force. The crane model has a maximum carrying capacity of 250 tons, where the force caused by the load by the catalog is 119 kPa.

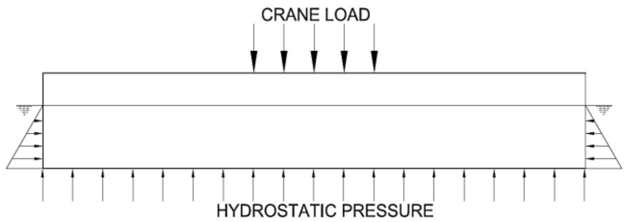

Figure 2. Deck crane load

The ship structure is one type of structure that has a very high complexity, it is necessary to simplify the analysis process by selecting several structures that are the main focus of the analysis process (Ertas et al., 2014). In this study, the simplification is done by modeling the construction between the two bulkheads, i.e. frame 18 to 21. As shown in Figure 3, the construction profile from frame 18 to frame 23.

In FEM modeling, to represent the actual structure, element selection needs to be done so that the element can represent the structure. In principle, element types can be categorized into three types, i.e. 1D elements, 2D elements, and 3D elements. 1D elements provide simplicity but lack their ability to model complex geometries. 2D elements and 3D elements provide more detailed results but require more computer time and memory in the modeling process

(Wai et al., 2013).

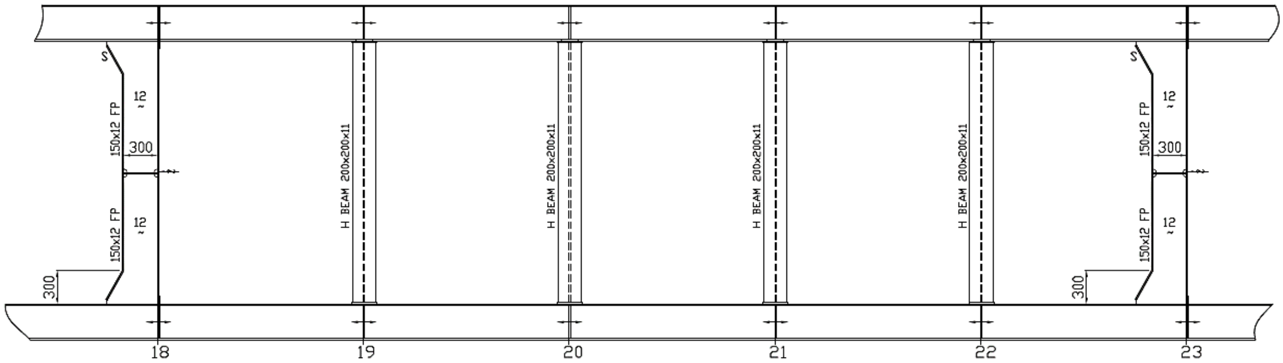

Figure 3. Construction profile frame 18-23

Elements in FEM are recognized by their shape, for example, elements can be straight lines, curves, triangles or quadrilaterals, tetra-hedrals, and so on. Line elements are the simplest elements that are made of two points. All line elements, both straight and curved, are called 1D elements, which have translational and rotational displacement functions (Berlioz & Trompette, 2010). Examples of 1D elements are beam elements and truss elements (Rugarli, 2010). 2D elements are surface elements with triangular or quadrilateral shapes (Berlioz & Trompette, 2010). Examples of 2D elements are 3-node triangular elements, 6-node triangular elements, and others (Rugarli, 2010). 3D elements are commonly used in volume meshes, derived from 2D elements, and used when the problem cannot be simplified (Wai et al., 2013). 3D solid elements only take into account translational displacements. In this study, the elements SHELL181 and BEAM189 are used to model the construction. SHELL181 is a 3D shell with linear element order. The BEAM 189 element is a quadratic three-node beam element in 3-D. Element SHELL181 is a 4-nodal element with 6 degrees of freedom at each node. Translation in the x, y, and z directions and rotations around the x, y, and z axes. Element SHELL181 is ideal for the analysis of thin to medium shell structures (ANSYS, 2011). SHELL181 accommodates the effect of transverse shear (Asadnia & Roddis, 2018). SHELL181 is applicable for sandwich materials with material and geometry properties similar to constructions used in the aviation industry (Banerjee et al., 2011). The BEAM189 element is suitable for analyzing thin/medium thick and thin beam structures. This element is a three-dimensional, three-node square bar element. Each node has 6 degrees of freedom and includes translations in the x, y, and z directions and rotations. BEAM189 is based on the Tymoshenko beam theory. The BEAM189 structure is made of 3 nodes (quadratic), used in the lattice frame (Jurčíková & Rosmanit, 2013).

Mesh density is one of the factors that determine the accuracy of results from modeling with the finite element method (Abubakar & Dow, 2013). The function of meshes in finite element modeling is to form structures in small elements or discrete objects. More elements are needed to improve the finite element solution (Logan, 2007). Mesh convergence determines how a model produces analyses that do not change with increasing number of elements. The response of a system including stresses and deformations will converge (unchanged and repeatable) as the element size decreases. Exact solutions increase with the number of elements. The process of re-meshing the element by ensuring the appropriate model and boundary conditions can make better convergence (Melosh, 1990). The smaller of element size, the model will be divided into smaller and make larger the number of elements. With the increase in the number of elements, the computer equipment for calculating the calculations increases, as a large computer capacity is required. The basic elements (uni-, bi-, or tri-dimensional) can be mapped in simple or complex geometries. The mapped mesh is easy to identify since it has all interior nodes with the same number of adjacent elements. The mapped mesh is usually defined in terms of quadrilaterals or hexahedrals (Lee et al., 2015). The meshed with Tri format should be 35% smoother than the Quad or mapped format (Lajarin et al., 2011). In this study, the mesh size used in modeling with mesh sizes 0.8, 0.7, 0.6, 0.5, 0.4, and 0.3 m. The mesh model in this study uses the shape-mapped method for areas and the shape-tri method for beams.

Determination of boundary conditions in modeling with FEM is done to represent the real conditions (Abdullah et al., 2023). In ship construction, boundary conditions in FEM modeling can use fixed boundary conditions placed at the bottom and ends of the structure as a support, this boundary condition functions to limit the area to be analyzed (Trihantoro et al., 2022).

Boundary conditions on ship construction in the form of plates with frames, it is not recommended to use simply supported boundary conditions or fixed boundary conditions, but in extreme conditions, the boundary conditions can be used in modeling (Lee et al., 2015). In the determination of boundary conditions on the barge construction with local analysis, it is necessary to refer to the DNVGL-CG-0127 Sec. 3 Part 3 rules (Khairunnisa et al., 2023). In this study, the boundary conditions refer to DNVGL-CG-0127 Sec. 3 Part 3 (DNVGL, 2015), the boundary condition can be seen in Figure 5.

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 5. Boundary conditions according to DNVGL-CG-0127 (DNVGL, 2015)

RESULTS AND DISCUSSION

The first step of finite element software simulation is to ensure that the results obtained are converged. in this study, convergence tests were carried out with mesh sizes of 0.8; 0.7; 0.6; 0.5; 0.4, and 0.3. as shown in the table Table .

The convergence process is principally carried out by changing the element size in the model and comparing the analysis results. The model has converged if the resulting analysis value has shown a stable value and has not changed significantly or the resulting curve has formed a straight line. The results of this test will be analyzed for allowable stress.

Vol. 17, No. 2, December 2023: page: 65-74
p - ISSN: 1978-886X
e - ISSN: 2614-641X

In Figure , the curve of the convergence test can be seen, the mesh size of 0.6 to 0.3 has a Von Mises Stress that starts to stagnate, and the difference in the modeling results is below 1%. This study uses a mesh size of 0.4 m based on convergence test results.

Table 5. Mesh size and number of elements

Mesh Size (m)	Number of Elements	Von Mises Stress (N/mm^2)
0.8	2701	154.1
0.7	2717	154.2
0.6	3492	157.3
0.5	6087	157.9
0.4	6833	158.0
0.3	8729	158.0

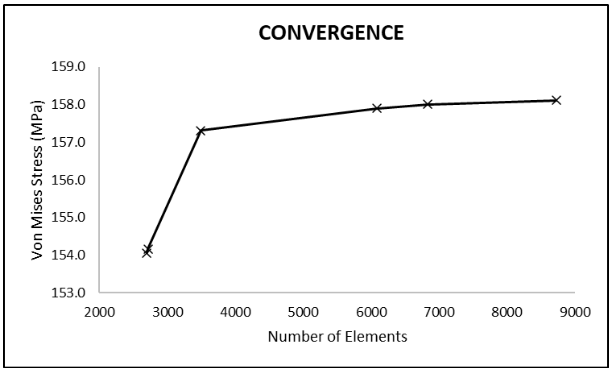

Figure 6. Convergence test results

The result of running software FEM, it was found that maximum stress at the connection between the bulkhead girder and the transverse frame (bracket), as shown in Figure .

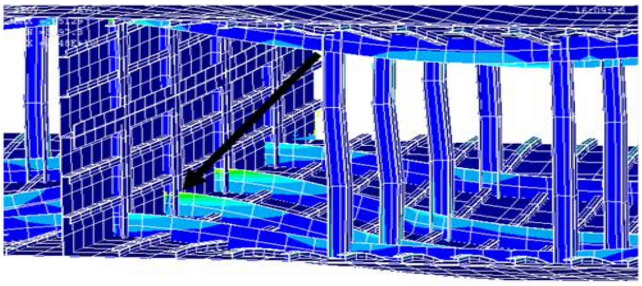

Figure 7. Location of maximum stress

This result is similar to research conducted by Pujikuncoro, et al. (2016); the critical location of the work barge structure is on the bracket between the watertight bulkhead and longitudinal bulkhead (Pujikuncoro et al., 2016).

For strength assessment using finite element analysis from Biro Klasifikasi Indonesia (BKI), the Von Mises Stress is less than equal to the nominal upper yield of the material, as shown in Equation (1) (Biro Klasifikasi Indonesia, 2022).

$$\sigma_v \leq R_{eH} \tag{1}$$

R_{eH} is the Nominal upper yield point of the material used (N/mm^2), the normal steel R_{eH} = 235 N/mm^2.

So, the Von Mises stress is less than equal to 235 N/mm^2, as shown in Equation (2).

$$\sigma_v \leq 235\ N/mm^2 \tag{2}$$

The von Mises stress is used to predict if a material will fracture or yield. This kind of stress is typically applied to ductile materials, such as metals. The von Mises stress is the equivalent stress or effective stress that is expected to occur in a ductile material (Jong & Springer, 2009). A material will yield, according to the von Mises yield criterion, if its von Mises stress under load is equal to or higher than its yield limit under simple tension.

Table 6. Stress results in each model

Model	Von Mises Stress (N/mm^2)
Model A	158.00
Model B	158.81
Model C	160.11

In Table , can be seen that all models have a stress below 235 N/mm^2. The results on all models have von Mises stress still below the permissible stress required by BKI. Model A which is the existing construction has a Von Mises Stress of 158 N/mm^2, while model B with a smaller longitudinal profile size L 85x85x8 mm, has a Von Mises Stress of 158.81 N/mm^2, in model C Von Mises Stress of 160.11 N/mm^2, with a longitudinal profile size of L 80x80x8 mm.

Reducing the size of the web plate and face plate in the longitudinal construction of the deck crane barge produces insignificant Von Mises Stresses with the same load. The deck crane barge construction in this study is still below the allowable stress with a reduction in longitudinal size up to size L 80x80x8 mm. In 2016, Salimah (2016) stated that the deck structure of the crane barge is designed to withstand more loads, so that the crane barge deck can withstand more loads.

CONCLUSION

In this research, the strength of the crane barge deck structure has been investigated numerically using the finite element method. Modeling with the assistance of finite element software can shorten and simplify the process of calculating the strength of ship construction. The existing construction (Model A) with a profile size of L 90x90x8 mm, resulted in a Von Mises Stress of 158.00 N/mm^2, while models B and C which have smaller web and face plates resulted in greater Von Mises Stress of 158.81 N/mm^2 and 160.11 N/mm^2. The reduction in the size of the face and web plates in the longitudinal section reduces the strength of the ship construction with higher stress values, but still below the permissible stresses required by BKI.

REFERENCE

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*, Tanjung Pinang, October 11-13, pp. 1-12.

Abubakar, A., & Dow, R. S., 2013. Simulation of Ship Grounding Damage using the Finite Element Method. *International Journal of Solids and Structures*, 50(5), pp. 623-636.

ANSYS, 2011. *ANSYS Mechanical APDL Element Reference*. https://www.yum-pu.com/en/document/read/7801539/mechanical-apdl-material-reference-ansys [Accessed December 15, 2023].

Asadnia, M., & Roddis, W. M. K., 2018. Modeling Out-of-Flatness and Residual Stresses in Steel Plate Girders. *Proceedings of the Annual Stability Conference Structural Stability Research Council*, Baltimore, April 10-13, pp. 1-20.

Banerjee, B., Chen, J., Das, R., & Kathirgamanathan, A., 2011. *Comparison of ANSYS Elements SHELL181 and SOLSH190*. https://www.semanticscholar.org/paper/Comparison-of-ANSYS-elements-SHELL181-and-SOLSH190-Banerjee-Chen/1d389ed822d80a997695ef1ed1a26a7cf1354a30 [Accessed December 15, 2023].

Berlioz, A., & Trompette, P., 2010. *Solid Mechanics using the Finite Element Method*. Hoboken: John Wiley & Sons Inc.

Biro Klasifikasi Indonesia, 2022. *Rules For Classification And Construction Part 1 Seagoing Ships Volume II Rules For Hull*. BKI, Jakarta.

DNVGL, 2015. *Class Guideline DNVGL-CG-0127 Finite Element Analysis*. DNVGL, Oslo.

El-Reedy, M. A., 2012. *Offshore Structures Design, Construction, and Maintenance*.

Houston: Gulf Professional Publishing.

Ertas, A. H., Alkan, V., & Yilmaz, A. F., 2014. Finite Element Simulation of a Mercantile Vessel Shipboard Under Working Conditions. *Procedia Engineering*, 69, pp. 1001-1007.

Firdaus, N., Djatmiko, E. B., Prastianto, R. W., & Ismail, M. F., 2021. Analisis Respon Gerak Floating Crane Barge untuk Decommissioning Struktur Lepas Pantai. *Wave: Jurnal Ilmiah Teknologi Maritim*, 15(1), pp. 31-44.

Jong, I., & Springer, W., 2009. *Teaching von Mises Stress: From Principal Axes to Non-Principal Axes*. Washington: American Society for Engineering Education.

Jurčíková, A., & Rosmanit, M., 2013. Recommendations for Numerical Modeling and Analytical Assessment of a Planar Steel CHS Joint. *VERSITA*, XIII, pp. 1-10.

Khairunnisa, Nugroho, P. N. A., & Abdullah, K., 2023. Stress Analysis of 91.5 Metre Coal Carrier Pontoon with Variations of Frame Distance. *Wave: Jurnal Ilmiah Teknologi Maritim*, 17(1), pp. 7-15.

Konecranes, 2019. *Konecranes Gottwald Cranes on Barge*. https://www.konecranes.com/discover/konecranes-gottwald-cranes-on-barge [Accessed December 15, 2023].

Lajarin, S. F., Magalhães, J. S. F., & Marcondes, P. V. P., 2011. Numerical and Experimental True Strain Assessment on Sheet Forming Using Mapped Versus Free Meshing. *Journal of Achievements in Materials and Manufacturing Engineering*, 49(2), pp. 323-330.

Lee, S. E., Thayamballi, A. K., & Paik, J. K., 2015. The Ultimate Strength of Steel Brackets in Ship Structures. *Ocean Engineering*, 101, pp. 182-200.

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

Melosh, R. J., 1990. Finite Element Analysis Convergence Curves. *Finite Elements in Analysis and Design*, 7(2), pp. 115-121.

Pujikuncoro, F. T., Zakki, A. F., & Yudo, H., 2016. Studi Analisa Kontruksi Deck Kapal Accommodation Work Barge Pada Fr 0-12 Akibat Penambahan Crane Berbasis FEM. *Jurnal Teknik Perkapalan*, 4(4), pp. 821–828.

Rugarli, P., 2010. *Structural Analysis with Finite Elements*. London: Thomas Telford Ltd.

Salimah, N., 2016. *Analisa Kekuatan Stuktur Deck Crane 15 Ton Pada Accomodation Barge Menggunakan Software Solidwork 2016*. Tugas Akhir. Politeknik Negeri Batam, Batam.

Soares, C. G., 2011. Ships and Offshore Structures Analysis and Design of Marine Structures. *Ships and Offshore Structures*, 6(1-2), pp. 1.

Trihantoro, A., Mulyanto, I. P., & Amiruddin, W., 2022. Analisa Kekuatan Struktur Deck Crane Kapal Tanker 6500 DWT Menggunakan Metode Elemen Hingga. *Jurnal Teknik Perkapalan*, 10(2), pp. 52-59.

Vukelić, G., & Vizentin, G., 2017. Common Case Studies of Marine Structural Failures. In: *Failure Analysis and Prevention*. London: IntechOpen, pp. 216.

Wai, C. M., Rivai, A., & Bapokutty, O., 2013. Modeling Optimization Involving Different Types of Elements in Finite Element Analysis. *Proceedings of the 2nd International Conference on Mechanical Engineering Research*, ICMER 2013, Kuantan, July 1-4, pp. 1-8.

Wulandari, A. I., Suardi, & Rusparyansyah, D. P. W., 2022. Analisis Kekuatan Memanjang pada Geladak Kapal Kontainer 409 TEU dengan Metode Elemen Hingga. *Wave: Jurnal Ilmiah Teknologi Maritim,* 16(1), pp. 17-22.

BLANK PAGE