Strength Analysis of the Deck Crane Barge Using the Finite Element Method

Kharis Abdullah[1*], Sumardiono [2], Hariyanto Soeroso[3]

{kharis.abdullah@ppns.ac.id[1*], dion@ppns.ac.id, hariyanto_soeroso@ppns.ac.id}
Department of Ship Building Engineering, Politeknik Perkapalan Negeri Surabaya, Surabaya, Indonesia[123]

Abstract. Barge is one type of ship that is widely used to transport various cargoes. One type of barge is a deck crane barge, barges with shear leg cranes or full rotating cranes. In the construction design, it is necessary to calculate the strength to avoid construction failure. One of the methods used to construction calculations is the finite element method. The finite element method is one of the most widely used numerical methods for engineering calculations. In this study, the strength of the deck crane barge was calculated using the finite element method. SHELL181 and BEAM189 elements were used in this study to simplified the modelling. The results of modeling with the finite element method produce that the maximum stress occurs at the connection between the bulkhead girder and the transverse frame (bracket), at Node 625. The construction strength of the deck crane barge model with a crane capacity of 250 tons (model 1) below the permissible stress, the value is 151.01 MPa. And the stress of the deck crane barge with a crane capacity of 350 tons (model 2) is 190.92 MPa, the stress exceeds the permessible stress.

Keywords: barge, crane, finite element method.

1 Introduction

In the marine industry, various types of transportation equipment and marine building facilities are developed to support existing industrial processes, such as ships, offshore construction, robot inspection etc. In the process of constructing a marine building such as an offshore structure, jetty, and various other facilities, it requires a tool for the transportation process and the constructions process. One type of transportation is barge which is a type of ship used to transport various goods and can be used as an offshore building construction facility. The barge used for construction of offshore building facilities is a deck crane barge. Deck crane barge is one of the facilities in the field of construction support in offshore facilities, where crane barges are offshore barges with shear leg cranes or full rotating cranes, crane barges are widely used because of their ability to install jackets into the frame with a tolerance of only 50 mm [1]. Not only that, the deck crane barge very flexible in its application and can be used anywhere on water, or as a backup or substitute for dock cargo handling equipment [2]. As shown in **Fig.** 1**.** [2], it is a deck crane barge using full rotating cranes.

ICSEDTI 2022, October 11-13, Tanjungpinang, Indonesia

DOI 10.4108/eai.11-10-2022.2326425

Fig. 1. Crane barge with full rotating cranes

In the design process of a construction, especially the construction of marine buildings, there is one step that must be done, it is the process of checking the strength. Where this stage is carried out to ensure that the structure and construction to be built has a structural strength that meets the requirements. This process is to avoid structural failure, where structural failure is one of the things that is avoided in construction design. Various factors of structural failure can cause damage to the ship so that it can affect operations [3]. Marine structures must comply with the design requirements in such a way that the probability of failure can be reduced to a minimum [4] . In addition, the strength of the structure of the ship or marine structure affects the durability of the structure itself. Ship structural durability can be defined as the ability of a structure to maintain its mechanical performance throughout its life. To produce a safe and reliable ship construction design, construction analysis is required under various static and dynamic loading conditions. Ship construction is one of the construction systems that is quite complex and complicated, so in general the analysis process is carried out by selecting several parts of the construction [5].

In calculating the strength of a structure can use empirical and numerical methods. At this time, the various methods are developed to facilitate the calculation of a construction or structure. One of them is the numerical method using the finite element method (FEM) which is one of the most widely used methods in the analysis of a structure. FEM is advantageous in its wide range of adaptability, e.g. its application to problems with heterogeneous materials, impeding complex geometric structures and various types of nonlinearities. In addition, the finite element method simplifies the analysis process. The calculation of the finite element method can be done by numerical simulation with a computer with the help of software. Examples of software for calculations using the finite element method are SAP 2000, NASTRAN, ANSYS, LS-DYNA, and others. By using the power calculation software is quite efficient and fast [6]. In this study, the strength of the deck crane barge structure will be discussed using the finite element method.

2 Model Details

In this research, the detailed analysis model of the crane barge deck structure is divided into two parts, the material-geometry model and constructions.

2.1 Material-geometri Model

In general, the material used in ship construction uses A 36 steel material. In this study, the deck crane barge uses A36 steel, where in general A36 steel has material properties as in Table 1, it can be seen that the Young Modulus value of A36 steel is 260 GPa, ultimate tensile strength 400 MPa, Poisson's ratio 0.26, and density 7850 Kg/m^3.

Table 1. Material properties of Grade A36 steel.

Material Properties	**Values**	**Unit**
Young Modules	260	GPa
Ultimate Tensile Strength	400	MPa
Poisson's ratio	0.26	
Density	7850	Kg/m^3

In this study, the deck crane barge has dimensions of an Length overall (LoA) of 54 m, a width (B) of 18 m, a height (H) of 3 m, and a draft (T) of 2 m, as shown in Table 2.

Table 2. Geometric properties of the barge.

Principal Dimension	**Values**	**Unit**
Length overall (LoA)	54	m
Width (B)	18	m
Depth (D)	3	m
Draft (T)	2	m

The crane barge design in this study can be seen in **Fig.** 2., where the deck crane barge has a fully rotating crane type, where this type of crane can rotate fully.

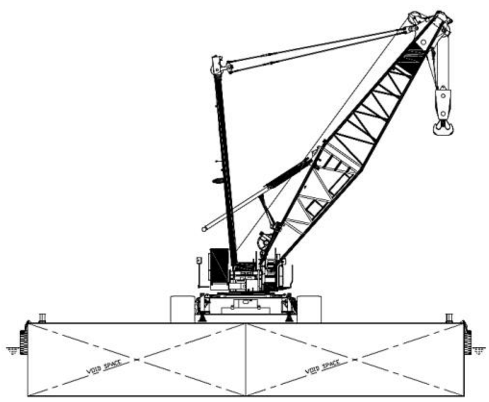

Fig. 2. Desain deck crane barge

2.2 Constructions

Construction on the deck crane barge is shown in **Fig.** 3**.** Cross section of the construction of the deck crane barge. Where the construction uses a longitudinal construction system.

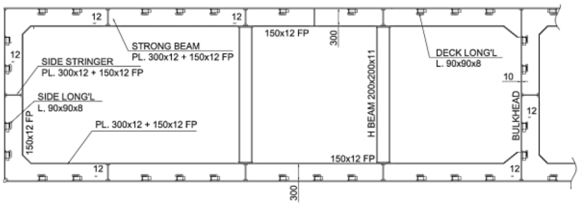

Fig. 3. Midship section port side view

The plate and profile sizes used in the deck crane barge construction are as shown in Table 3, where the plate and construction sizes refer to the class society rules Biro Klasifikasi Indonesia.

Table 3. Plate and Profile Construction

Part	Plate/Profile (mm)
Bottom Plate	12
Bottom Longitudinal	L 90 x 90 x 8
Bottom Transversal	PL. 300x12 + 150x12 FP
Side Plate	10
Side Longitudinal	L 90 x 90 x 8
Side Stringer	PL. 300 x 12 + 150 x 12 FP
Deck Plate	12
Deck Longitudinal	L 90 x 90 x 8
Deck Beam	PL. 300 x 12 + 150 x 12 FP
Pillar	H 200x200x1

3 Finite Element Method

The finite element method is a numerical method that is widely used in engineering work processes, such as structural strength, fatigue, material failure and the others. In general, calculations using the finite element method are assisted using software assistance. Modeling in the finite element method software has several stages. These stages must be done correctly so that the model can be run or can be processed calculations assisted by a computer [6], several stages in using finite element software in general there are three stages, that is:

1. Preprocessing
   At this stage the problem definition is carried out by describing the element type, geometry and mesh size.
2. Solution

This stage is the second stage where the application of load or force, boundary conditions, and settlement or solving is carried out.

3. Postprocessing
   The last stage is reading the results of the processes that have been carried out, such as stress, displacement, contour diagrams and others.

3.1 Forces

In this study, the load acting on the construction is the load of the hydrostatic force and the load from the crane weight. The force load applied to the crane barge is in conditions when the water is calm, where there is no intervention from sea waves. Hydrostatic pressure occurs at a water level of two meter and the force load caused by crane pressure is in accordance with the specifications set by the existing crane maker or catalogue. As shown in **Fig.** 4, the load on the crane barge consists of the weight of the crane and the hydrostatic force.

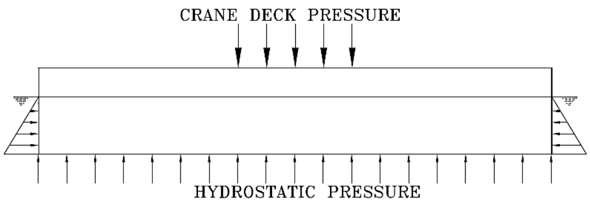

Fig. 4. Pressure on deck crane barge

The existing design has two alternative uses of cranes, where the crane model 1 has a maximum carrying capacity or safe working load of 250 tons, where the force caused by the existing load is in accordance with the catalog of 108.4 kPa, and the other alternative crane model uses a type of crane capable of carrying 350 tons, with a force load of 150 kPa, as shown in Table 4.

Table 4. Crane Forces

Model	Maximum Capacity (Ton)	Forces (kPa)
Crane Model 1	250	108.4
Crane Model 2	350	150.0

3.2 Model Description

Ship construction is one of the most complex constructions, so simplification is needed by selecting several constructions to be analyzed [5]. In this study, modeling was carried out on the between two bulkhead construction, that is from frame 13 to frame 18. As shown in **Fig.** 5, the construction profile frame 13 to frame 18

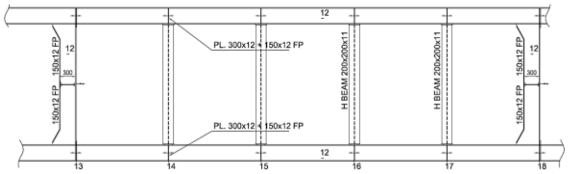

Fig. 5. Construction Profile Frame 13-18

3.3 Element

The selection of elements is one of the important stages in finite element modeling so that it can represent the actual condition of the structure. In general, the types of elements in finite element software can be categorized into two types, it is 2D elements and 3D elements. These elements consist of point elements, line elements, area elements, and solid elements. Existing elements can be combined according to modeling needs. As we know, ship construction is a complex construction, so the use of various types of elements can be used in modeling. In this study, 2D elements are used with a combination of line elements as beams and shell elements as plates and ANSYS software was used to assist the process of calculating ship structures.

- Shell Element

  Element SHELL181 is suitable for analysis of thin to medium shell structures. This is a 4 nodal element with 6 degrees of freedom at each node.Shifts in the x, y, z directions and rotations around the x, y, z axes [7]. As we can see in **Fig.** 6, shell geometry of SHELL 181. This element can accommodate the effect of transverse shear [8]. On the other hand, SHELL 181 can be used for sandwich materials with material properties and geometries similar to constructions used in the aviation industry [9].

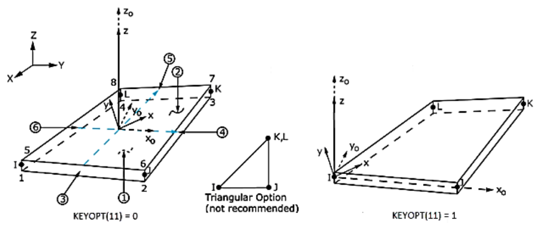

Fig. 6. SHELL181 Geometry

- Beam Element

  The BEAM189 element is suitable for analysis of thin to moderately dull/thick beam structures. This element is a square bar element with three nodes in three dimensions. By default, each node has 6 degrees of freedom. This includes translations in the x, y, and z directions and rotations about the x, y, and z directions. This element is based on Tymoshenko beam theory including shear strain effects. This element provides options for unlimited and constrained warpage of the cross section [7]. In **Fig.** 7, it can be seen that the geometry of the BEAM189 element beam shows that the element beam has three axes, x, y and z. Element beam 189 can be used in the lattice frame model because the BEAM 189 structure is made of 3 nodes (quadratic) [10].

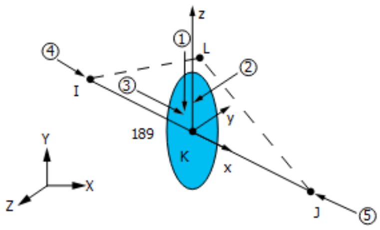

Fig. 7. BEAM189 Geometry

3.4 Meshes

Analysis using the finite element method is one technique that provides an approximate solution, and its accuracy where the modeling results depend on a number of factors including mesh density [11]. The function of the mesh in the modeling process with finite elements is to form a structure into small elements or discretized bodies. To improve the solution from finite elements, we need more elements [6]. In modeling with the finite element method, convergence is one of the important stages in the analysis process. Mesh convergence determines how a model produces an analysis that does not change with the number of elements, so it can be said that how many elements are needed in a modeling. The response of a system including stress and deformation will converge (unchanged and repetitive) as the element size decreases. On the other hand, convergence is very important in the decision process, as shown in **Fig.** 8, it can be seen that the coverage of the exact solution increases with the number of elements present [6].

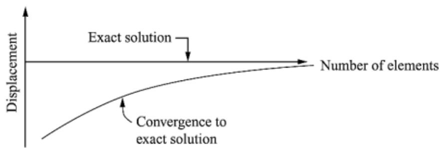

Fig. 8. Convergence to the exact solution for displacement

The convergence curve in finite element analysis defines the relationship between grid intervals and analysis accuracy. The remeshing process can make better convergence by ensuring the appropriate model and boundary conditions [12]. The smaller the element size, the larger the number of elements, because the model will be divided into smaller ones. On the other hand, with the increase in the number of elements, the equipment for computing the existing calculations increases. Basic elements in FEM (uni-, bi- or tri-dimensional) can be mapped in simple or complex geometries. The mapped nets are easily identified because they have all their interior nodes with the same number of adjacent elements. A mapped mesh is typically defined in the quadrilateral or hexahedral [13]. The meshed sheet with Tri format should be 35% smoother than the Quad or mapped format [14]. Can be seen in **Fig.** 9, mapped mesh on the existing modeling structure.

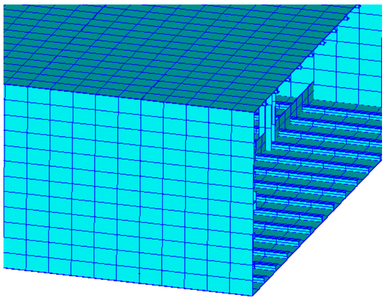

Fig. 9. Mapped mesh with quadrilateral

3.5 Boundary Conditions

It is very necessary to determine the boundary conditions so that the modeling is in accordance with the existing conditions. Modeling boundary conditions in ship construction which is a plate that has continuous reinforcement is not recommended to use simply supported or fixed boundary conditions, but in extreme conditions fixed boundary conditions can be used in the modeling [13]. research conducted by trihantoro 2016, states that the condition of the clamping boundary is placed at the bottom and the end of the structure as a support. This limit is given with the aim of limiting the area to be analyzed [15]. In this study, construction modeling is limited to the construction between the frame 13 to frame 18. The analysis is static so that the truncated frame is analogous to a clamped condition, where: displacement and rotation have been set to zero. This limitation is applied to obtain a situation similar to the condition when the crane barge is working. The load put on on the structural model, in the form of crane pressure on the deck of the ship structure, and then the hydrostatic loads to bottom, and right-left side of the ship structure.

3.6 Analysis

After doing the modeling and meshing process, the next process is the analysis process using materials and techniques in accordance with existing procedures. As shown in **Fig.** 10, the results of the running model for the existing deck crane barge construction can be seen. Where in the plot results for Von Mises Stress. Equivalent stresses (also called von Mises stresses) are often used in design work because they allow any arbitrary three-dimensional stress state to be represented as a single positive voltage value. Equivalent stress is part of the maximum equivalent stress failure theory which is used to predict yielding in ductile materials.

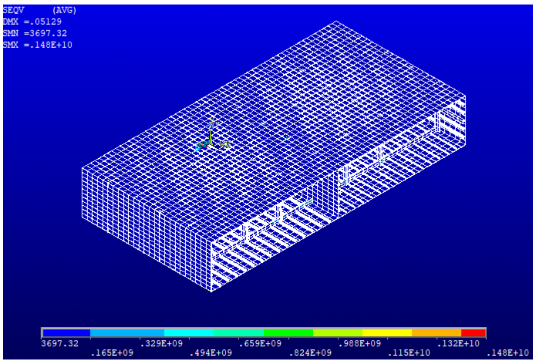

Fig. 10. The plot illustrates the von Mises stress

4 Results

Convergence is quite important in the finite element modeling process, so that the resulting error is getting smaller. In this study, the mesh sizes used were 0.8 m, 0.7 m, 0.6 m, 0.5 m, and 0.4 m. In **Fig.** 11, it can be seen that the mesh size of 0.6 to 0.4 has a Von Mises Stress that starts to stagnate. The modeling results with a mesh size of 0.6 m is 157.31 MPa, a mesh size of 0.5 is 157.97 MPa, and a mesh size of 0.4 m is 158.01 MPa. The error of the modeling results is below 1%, so that the next modeling uses a mesh size of 0.4 m.

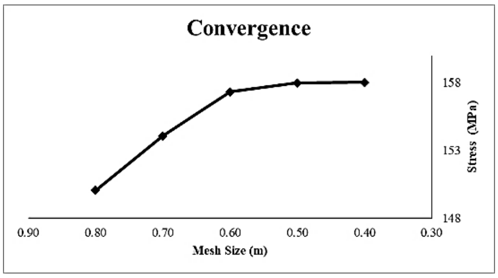

Fig. 11. The mesh convergence

The maximum stress that occurs in this modeling at the connection between the bulkhead girder and the transverse frame (bracket), at Node 625. This high stress point is similar to research conducted by Pujikuncoro, 2016, which states that the critical location of the work barge structure due to the addition of a crane is located on the bracket between the watertight bukhead and longitudinal bulkhead [16]. As shown in **Fig.** 12, the position of the maximum stress occurs.

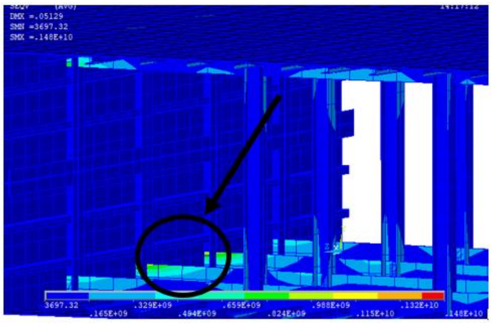

Fig. 12. Maximum stress location

In the rules of the Indonesian Classification Bureau (Biro Klasifikasi Indonesia) Volume II Rules for Hull, it states that "the normal strength of ship hull construction steel is hull construction steel with a minimum upper yield value of ReH 235 N/mm^2 and a tensile strength of Rm 400 - 520 N/mm^2" [17]. In this study, the material is assumed to have a ReH value of 235 N/mm^2. So that the existing permessible stress is

$$permessible\ stress = \frac{180}{k}\ N/mm^2 \tag{1}$$

$$where.\ k = \frac{235}{ReH} \tag{2}$$

So, the permessible stress is 180 MPa.

$$permessible\ deformation = 0.4\ \%\ l \tag{3}$$

$$where.\ l\ is\ unsupported\ span.\ in\ this\ case\ the\ l\ is\ 1830\ mm$$

So, the permessible deformation is 7.32 mm

In Table 5, it can be seen the results of the calculations that have been carried out where in the first model, the deck crane barge with a maximum capacity of 250 tons (crane model 1), still has a maximum stress below the permessible stress, while in the second model crane exceeds the permessible stress that has been set, so that the construction of a deck crane barge with a crane capacity of 350 tons (crane model 2) must be changed in its construction or structure in order to be able to withstand the existing load. The construction of deck crane barge can withstand more loads, because the deck structure is designed to withstand more loads [18].

Table 5. Result of Von Mises Stress and Deformation

Model	Von Mises Stress (MPa)	Deformation (mm)	Accepted
Crane Model 1	151.01	0.81	V
Crane Model 2	190.92	1.07	X

5 Conclusion

In this study, the condition of the ship structure (deck crane barge) was investigated numerically. Where the finite element method is used to calculation the strength of the ship's structure. Modeling with finite element software can shorten the process of calculating the strength of construction and failure that will occur. In the first crane model the strength of the existing structure is still below the permessible stress that has been set, while in the second model has the maximum stress greater than the permessible stress. Significant results indicate that greater structure profiles and construction is required to windstand load a deck crane barge with a capacity of 350 tons. However, further study of the fatigue strength assessment of barge structures is required.

References

[1] Mohamed A. El-Reedy, *Offshore Structures Design, Construction and Maintenance*. Gulf Professional Publishing, 2012.

[2] Konecranes, "Konecranes Gottwald Cranes on Barge," 2019.

[3] P. Taylor and C. G. Soares, "Ships and Offshore Structures Analysis and Design of Marine Structures," no. January 2015, pp. 1–2, 2011, doi: 10.1080/17445302.2011.546686.

[4] G. Vukelić and G. Vizentin, "Common Case Studies of Marine Structural Failures," *Fail. Anal. Prev.*, 2017, doi: 10.5772/intechopen.72789.

[5] A. H. Ertas, V. Alkan, and A. F. Yilmaz, "Finite element simulation of a mercantile vessel shipboard under working conditions," *Procedia Eng.*, vol. 69, pp. 1001–1007, 2014, doi: 10.1016/j.proeng.2014.03.082.

[6] D. L. Logan, *A first course in the finite element method*, 4th ed. Thomson, 2007. doi: 10.1016/0168-874x(87)90008-4.

[7] T. D. Canonsburg, *ANSYS Mechanical APDL Element Reference*, no. November. 2013.

[8] M. Asadnia and W. M. Kim Roddis, "Modeling out-of-flatness and residual stresses in steel plate girders," *Proc. Struct. Stab. Res. Counc. Annu. Stab. Conf. 2018*, no. June, 2018.

[9] B. Banerjee, "Comparison of ANSYS elements SHELL181 and SOLSH190," no. September, 2014, doi: 10.13140/RG.2.1.1406.3445.

[10] A. J. Jurčíková and M. Rosmanit, "Recommendations for Numerical Modeling and Analytical Assessment of a Planar Steel Chs Joint," *VERSITA*, vol. Vol. XIII, no. February, 2013, doi: 10.2478/tvsb-2013-0009.

[11] A. Abubakar and R. S. Dow, "Simulation of ship grounding damage using the finite element method," *Int. J. Solids Struct.*, vol. 50, no. 5, pp. 623–636, 2013, doi: 10.1016/j.ijsolstr.2012.10.016.

[12] F. Elements and E. Engineering, "Finite element analysis convergence curves," vol. 7, pp. 115–121, 1990.

[13] S. E. Lee, A. K. Thayamballi, and J. K. Paik, "Ultimate strength of steel brackets in ship structures," *Ocean Eng.*, vol. 101, pp. 182–200, 2015, doi: 10.1016/j.oceaneng.2015.04.030.

[14] S. F. Lajarin, J. S. F. Magalhães, and P. V. P. Marcondes, "Numerical and experimental true strain assessment on sheet forming using mapped versus free meshing," vol. 49, no. 2, pp. 323–330, 2011.

[15] A. Trihantoro, I. P. Mulyatno, and W. Amiruddin, "Analisa Kekuatan Struktur Deck Crane Kapal Tanker 6500 DWT Menggunakan Metode Elemen Hingga," *J. Tek. Perkapalan*, vol. 10, no. 4, p. 52, 2022, [Online]. Available: http://ejournal3.undip.ac.id/index.php/naval

[16] F. T. Pujikuncoro, A. F. Zakki, H. Yudo, D. T. Perkapalan, F. Teknik, and U. Diponegoro, "Studi Analisa Kontruksi Deck Kapal Accommodation Work Barge Pada Fr 0-12 Akibat Penambahan Crane," *J. Tek. Perkapalan*, vol. 4, no. 4, pp. 0–7, 2016.

[17] B. K. Indonesia, "Rules for Hull (Pt.1,Vol.II) 2019 ed.," *Rules Hull 2019 ed.*, vol. II, p. Page 20–7, 2019.

[18] N. Salimah, S. W. Satoto, and N. Yuniarsih, "Analisa Kekuatan Stuktur Deck Crane 15 Ton Pada Accomodation Barge Menggunakan Software Solidwork," *J. Tek. Mesin*, 2016, [Online]. Available: https://103.209.1.145/uploads/207029-20171106071110.pdf