FEM Structural Analysis for Ship's Beam Modification: A Case Study

Adrian Popa
"Mircea cel Batran" Naval Academy, Romania
adrian.popa @anmb.ro

Mihaela-Greti Manea
"Mircea cel Batran" Naval Academy, Romania
greti.manea@anmb.ro (corresponding author)

Marian-Valentin Ristea-Komornicki
"Mircea cel Batran" Naval Academy, Romania
marian.ristea@anmb.ro

Received: 20 May 2024 | Revised: 10 June 2024 | Accepted: 24 June 2024

ABSTRACT

Classification is mandatory for all seagoing ships engaged in international trade worldwide, with regulations that require the strength of the hull construction to be confirmed. Upon completion of the ship's construction or during the execution of repair works in shipyards (reconversion works, modification of panels, replacement of equipment, etc.), the strength structure of the ship may be affected and the classification society must certify that it meets the requirements. Replacing old equipment sometimes implies some variations in overall dimensions, which may impose some changes (adjustments) on the layout of structural panel elements. This study applied the Finite Element Method (FEM) considering the von Mises theory in a case study of a beam modification with the support of the ANSYS software. This study used the reverse engineering method to investigate an already implemented solution.

Keywords-structural strength; von Mises stress; CAD model; finite element analysis; ANSYS

I. INTRODUCTION

Ship classification societies supervise and verify the design processes, the construction of new ships, and the maintenance and repair of ships in operation. Regulations require that hull construction must be strong enough to withstand all the stresses to which it is normally subject, and the strength must be demonstrated by providing proof of design calculations and a classification certificate or declaration from an approved classification society. The rules of a classification society contain requirements, standards, and norms for the dimensioning and construction of seagoing ships. Ship structural analysis has become a major part of marine engineering and provides essential support for design, construction, and repair projects. Structural analyses of ships (hull girder, longitudinal or transverse beams, and various panel types) are performed to investigate their safety and capacity, as ships are exposed to high risks of structural damage during voyage and/or harbor operations.

Ship structures have very complex load conditions, due to waves, cargo, onboard equipment, and other special loads, requiring local and global strength assessment on equivalent 1D and full 3D numerical models. According to the rules of shipbuilding classification societies and the recommendations of the International Ship and Offshore Structures Congress (ISSC) Proceedings, equivalent quasi-static design load approaches are used for very low-frequency components, and dynamic non-linear analyses are used for vibration, impact, and explosion extreme loads. The structural analysis for evaluating the strength of ships considers both simple (stretching - compression, bending, twisting, shear) and compound stresses (resulting from the combination of at least two simple stresses). It is necessary to analyze both statically (i.e., slow and with negligible variations in intensity) and dynamically applied loads (either by sudden application of loads or by loads with varying intensity over time).

Due to the large amount of data to be processed, specific computers and software are used to quickly explore a wider spectrum of variants before making design, construction, and/or structural modification decisions. This reverse engineering study used ANSYS [2], which contains a high-performance Finite Element analysis (FEM) and design module that can be used in marine engineering and naval architecture projects. FEM is a numerical method used to solve engineering problems that allows the analysis of physical phenomena to be described using mathematical models consisting of differential equation systems with initial conditions and limits [3]. The deformation of solid bodies represents only a part of the applicative potential of numerical modeling using FEM.

This study aims to highlight the advantages of using ANSYS software to apply FEM analysis considering the von Misses theory on a ship beam modification, which was necessary to permit the lifting operation of equipment. 3D Computer-Aided Design (CAD) models and the von Mises stress distribution are presented for the original (unmodified) and the modified beam. This study is a reverse engineering study using an ANSYS structural module, considering an already implemented modification onboard a ship.

The structural analysis of ships is a topic of interest regarding the ship beam (hull girder), various floors or beams (longitudinal, transverse), or other critical areas of the ship's strength. Traditional ship structural design follows the rule books of classification societies. Modern ship structural design involves initially the structural main elements according to the class rules, followed by a direct FEM structural strength analysis and assessment using various strength criteria (admissible value to the yielding von Mises stress limit, buckling, plastic capability, ultimate strength, long-term fatigue). The design includes structure optimization by various criteria, according to the rules of shipbuilding classification and the recommendations of the ISSC Proceedings. Structural analysis programs and tools can be used to verify classification rules and accurate design in small details [4]. In [5], a rule-based methodology and software-based analyses were applied to analyze some of the mechanical properties of the ship, such as total deformation, stress-strain distribution, von Mises stress, fatigue, etc. This study provided some guidance for further improvements in ship structural design. In [6], the advantages of using FEM analysis were highlighted, demonstrating that using class manuals to estimate the ice load capacity of a ship's structural elements provided satisfactory results only after reducing the numerical coefficient. The consequences of the analysis were the changing frame space, the thickness of plating, the depth of stringers, and the area of application of the load.

The general concept of beam load and the problems related to beams and frames are significant in various engineering fields. In [7], the ship was understood as a beam resting on an elastic environment (sea waves) and subjected to static and dynamic wave loads. Regarding the strength of the ship under various stress types, an empirical formula was proposed in [8] to predict the ultimate strength of the initially deflected plate subjected to combined longitudinal compression and lateral pressure using Analysis of Large Plated Structures / Ultimate Limit State Assessment Program (ALPS/ULSAP). The problem of evaluating the ultimate strength of stiffened panels in ship hull structures that are subjected to combined uniaxial thrust in-plane and out-of-plane bending moments was examined in [9]. In [10], a practical method was proposed to estimate the extreme value distribution of von Mises stress for structural strength evaluations of ships, derived using approximations that reflect realistic stress conditions. This method was based on an asymptotic approximation that can be easily calculated in a way similar to the conventional linear statistical prediction. Complex subjects, such as buckling [11, 12], damage caused by collisions [13, 14], material qualities and properties, welding technology [15], corrosions, cracks and fatigue of materials [16, 17], and structural response of the critical section of the ship under hydrodynamic and hydrostatic sea loads [18], have attracted the interest of researchers to use finite element analysis methods and various types of software.

II. THEORETICAL CONSIDERATIONS

In general, the boundary state of a strength element is the stress condition leading to malfunction or failure. Examples of limit (boundary) states are numerous in the strength of materials: boundary states of total failure, fracture/rupture for brittle materials, plastic material behavior for ductile materials, boundary states of functional failure, vibration, corrosion, and buckling. The external loads on a body produce different states of stress and deformation at its points, which must be compared with the limited state of stress or deformation. Since the limit state of stress is determined experimentally at tensile test, a mono-axial, homogeneous, and uniform tension state, it is necessary to establish a criterion where any stress state can be compared with the limit state of simple stretching, which may be the elasticity limit, flow state, or breaking state. The hypothesis that a factor (normal stresses, tangential stresses, specific lengthening, specific slides, and specific deformation energy) is decisive in reaching the limit state and that its limit value is that of a simple tensile test constitutes a breaking/failure hypothesis, and the theory developed is called resistance theory [19]. Analysis of the tension states that various types of stresses incur in the volume of a solid body helps to establish how the body will behave under the action of its loads. One of the main purposes of this analysis is to estimate the level of charge at which it becomes possible to cease the stable and reversible behavior of that body and its eventual decommissioning. In the case of simple loads, it is relatively easy to determine when the materials no longer bear, under the correct conditions, the loads they are intended to take, that is, they "fail", because they reach a maximum admissible limit (usually called admissible resistance of the material to the given stress), by the highest tension produced in a material. It is much more difficult to establish a dangerous level of loads in cases where the analyzed body must take several simple stresses at once, that is, subjected to compound stresses. It is virtually impossible to perform mechanical tests for the wide range of possible stress states, which often have a high degree of complexity. Even for the study of a single material, many trials would be necessary on complicated construction machines and with numerous and difficult-to-execute specimens. These reasons justify why it is necessary to approximate limit states and investigate the degree of danger that the compound stress poses to the integrity of the studied body with the dangerous levels of simple mono-axial stress. More specifically, the admissible resistance established by the tensile test can be used in the calculation of plane or spatial stress states [20]. Within the domain of mechanics, it is essential to predict how materials respond to stress (σ). Several criteria are used, but one of the most comprehensive is the von Mises criterion, also known as the maximum distortion energy theory used for ductile materials. This theory is based on the concept of distortion energy (W), which is the internal deformation energy stored within a stressed material. Von Mises proposes that the point of irreversible plastic deformation (ε_y) occurs when the distortion energy reaches a critical value. Interestingly, this critical value can be determined by a simple uniaxial tensile test, in which a material sample experiences a single pulling force (σ_u).

The elegance of the von Mises criterion lies in its ability to predict yielding under complex stress scenarios (bending or twisting) using data from the basic tensile test. This is achieved by the following formula, which considers the various stress components in the material's stress tensor (Cauchy stress tensor, σ) and condenses them into a single scalar value called the von Mises stress (σ_v):

$$\sigma_v^2 = \frac{1}{2}[(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6(\sigma_{23}^2 + \sigma_{31}^2 + \sigma_{12}^2)] \quad (1)$$

If the calculated von Mises stress surpasses the yield strength (σ_y) obtained from the tensile test, the material is likely to yield.

III. MAIN STEPS OF FEM NUMERICAL ANALYSIS

FEM is a multidisciplinary method based on knowledge from three areas [3]:

- Mechanics of structures, including theory of elasticity, strength of materials, theory of plasticity, dynamics of structures, etc.
- Numerical analysis, including approximate methods, solving systems of linear algebraic equations, eigenvalue problems, etc.
- Applied computer science, dealing with the development and implementation of large computer programs.

Due to the computational possibilities it offers, FEM is widely used in assisted design and can be applied to systems with complex geometry and complicated parameter distributions. Simplistically speaking, if there is a structural 3D FEM model of the system to be studied, some aspects can be successfully studied, such as static structural analysis to determine displacements and stresses under the static liner and nonlinear conditions, and dynamics to determine the response of the structure to tasks that vary randomly over time until fault, fatigue, buckling, flow and failure analysis. Modeling with ANSYS implies some essential aspects:

- Geometry is the ANSYS workspace for creating 2D or 3D models that need to be further processed.
- Meshing allows geometries to be broken down into small polygons so that each can be processed separately to generate the results (the finer the mesh created, the better the results, but excessive network refinement can lead to long processing times).
- Configuring boundary conditions helps the software determine how each surface will behave when in contact with another.
- Calculation execution is the part where the solver uses complex mathematical equations to generate the results.
- Results is an ANSYS post-processor that visualizes the simulation outputs and analyzes how the system will behave in real-life scenarios.

IV. CONTRIBUTIONS AND ANALYSIS - CASE STUDY

This study was inspired by a structural modification onboard a ship. In the initial situation, due to new regulations, a scrubber pump was installed on a ship just beneath the main frame. Due to maintenance necessities, the pump drive had to be replaced, but there was no space available.

A. Problem to Solve

The maintenance team decided to modify the mainframe onboard as shown in Figure 1.

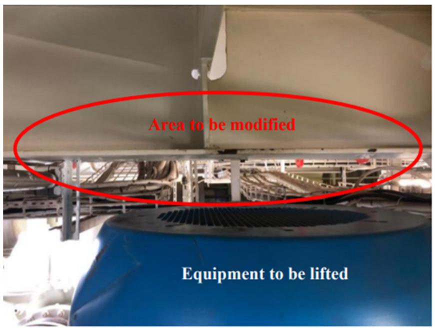

Fig. 1. Area to be modified - onsite picture.

The beam to be modified had dimensions W = 254 mm (10"), H = 343 mm (13.5"), T = 25.4 mm (1"), as shown in Figure 2.

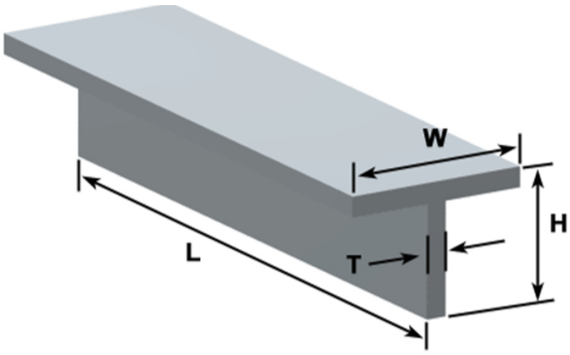

Fig. 2. Beam to be modified.

B. Design of the Original State

In the first step, it is necessary to study the initial beam without modifications. The 3D model of the original beam was designed using AutoCAD, as shown in Figure 3. Figure 4 also shows its unmodified state.

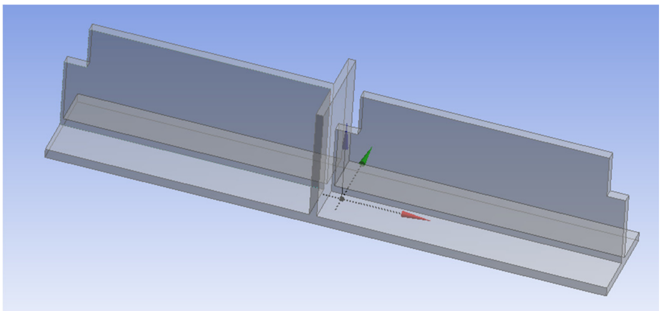

Fig. 3. Unmodified beam 3D model and coordinate system.

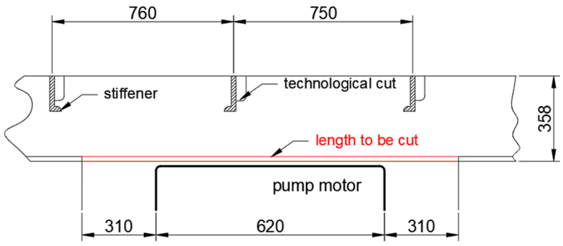

Fig. 4. Original state (unmodified beam).

C. Constructive Solution

Figure 5 shows the 3D CAD model of the modified beam. To carry out lifting operations in the area, it was necessary to raise the beam by 190 mm, as shown in Figure 6. The solution consisted of cutting a portion of the beam on a length of 1240 mm and reconfiguring the geometry with the help of two 254×25.4 mm (10×1") side flat bars, inclined by 45° (marked red in Figure 6). Further reinforcement was also required in the approximate central region with a 254×25.4 mm (10×1") vertical flat bar (marked blue in Figure 6), positioned 100 mm from the original stiffener. The stiffeners were extended to a total height of 358 mm (marked green in Figure 6).

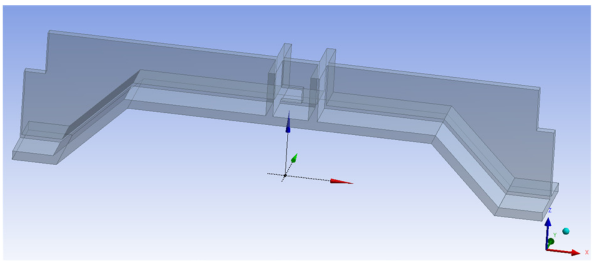

Fig. 5. Modified beam 3D model and coordinate system.

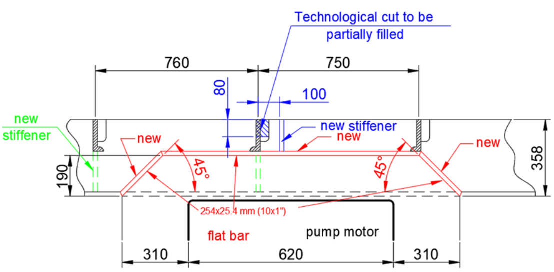

Fig. 6. Constructive solution (modified beam).

D. Structural Analysis of the Unmodified Beam

For the unmodified beam, the mesh consisted of 10595 nodes and 5084 elements, as shown in Figure 7. Figure 8 presents the loads and boundary conditions for the unmodified beam. Equivalent stress (von Mises stress) was considered for the post-processor results of the ANSYS software. The von Mises stress distribution is presented in Figure 9.

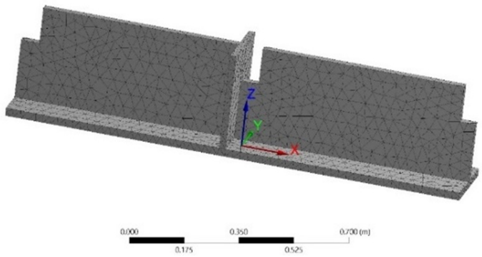

Fig. 7. Unmodified beam mesh.

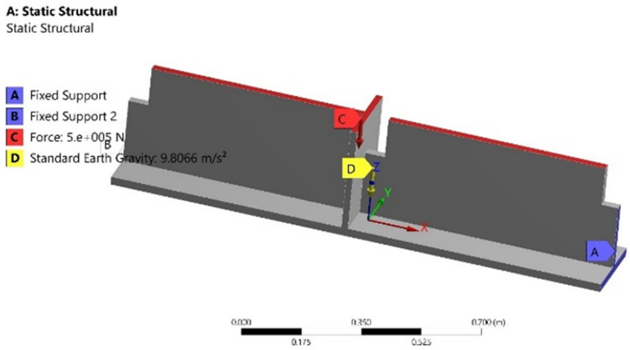

Fig. 8. Loads and boundary conditions for the unmodified beam.

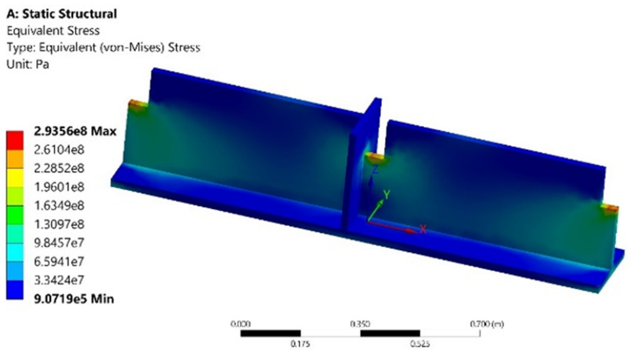

Fig. 9. Von Mises stress distribution for the unmodified beam.

The maximum value of von Mises stress is 293.56 N/mm^2. This is the value that this study tries to reach as the maximal von Mises stress after the beam modification.

E. Structural Analysis of the Modified Beam

For the modified beam, the mesh consists of 10478 nodes and 5065 elements and is shown in Figure 10. Figure 11 presents the loads and the boundary conditions for the modified beam.

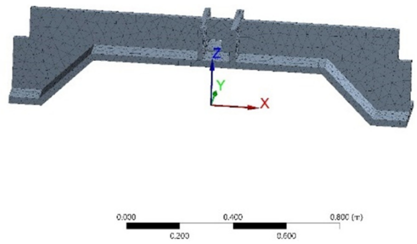

Fig. 10. Modified beam mesh.

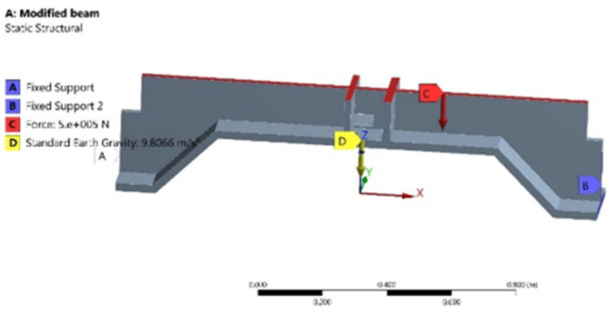

Fig. 11. Loads and boundary conditions for modified beam.

The post-processor results of the ANSYS software considered the equivalent stress (von Mises stress). Figure 12 presents the von Mises stress distribution, with its maximum value of 258.69 N/mm^2.

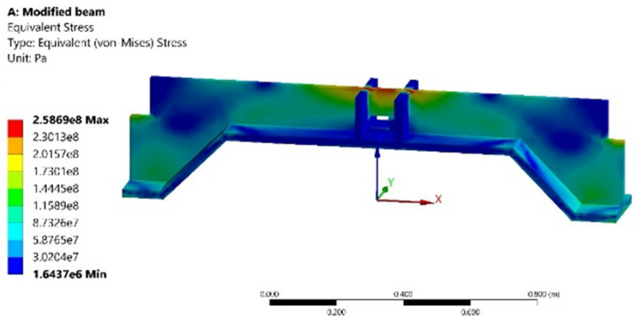

Fig. 12. Von Mises stress distribution for the modified beam.

V. CONCLUSIONS

Maintenance is an activity that keeps equipment, machinery, installation, or system running according to the designed functionalities. Regardless of whether it is a small machine or a large structure, efficient maintenance can help with prolonged life and favorable outcomes. From time to time, at intervals specified by class societies, it is necessary not only to repair equipment, but also to replace it with newer and better-performing ones. To replace equipment, it is necessary to access it so that it can be dismantled and removed from the ship. Sometimes, the overall dimensions of the new equipment require a larger space than the old one. Therefore, situations requiring geometric or dimensional changes in the ship's structure are not uncommon. Of course, these modifications must be made in such a way as to not affect the general and local strength of the ship and to continue to comply with the safety requirements of the class societies. This study was inspired by a previously implemented modification onboard a ship, where the geometry and dimensions of a beam were modified to lift a piece of equipment.

The subject was approached from the perspective of reverse engineering and accepted the presumption that the ship's representatives already had all the approvals for the modifications that were made. This study used ANSYS software in terms of von Mises stress theory, which is different from the standard way of work, hoping that this method represents a feasible alternative for structural analysis that is faster, cheaper, and as secure as the standard methods. To lift the equipment as it was necessary for maintenance reasons, the following modifications were performed:

- Cut a length of 1240 mm from the beam.
- Reconfigure the geometry to raise the beam by 190 mm.
- Make some reinforcement where necessary through 254×25.4 mm (10×1") flat bars.

The FEM analysis showed that by applying a 500 KN force on the initial and modified models of the beam, the maximum stress on the initial beam was 293.56 N/mm^2 and the maximum equivalent stress for the modified beam was 258.69 N/mm^2. The selected concentrated force of 500 kN was only a testing load to compare the modified version of the beam with its initial design. The modified beam is capable of taking the same loads as the initial beam, so the constructive modification does not affect its structural state and can be done safely. Since the modification was already implemented, it can be concluded that this method can be used to design and implement structural modifications.

REFERENCES

[1] European Parliament, *Directive (EU) 2016/1629 of the European Parliament and of the Council of 14 September 2016 laying down technical requirements for inland waterway vessels, amending Directive 2009/100/EC and repealing Directive 2006/87/EC*, vol. 252. 2016.

[2] "Ansys Software Training Center." https://www.ansys.com/training-center.

[3] J. Whiteley, *Finite Element Methods*. Cham, Switzerland: Springer International Publishing, 2017.

[4] Md. S. Islam, Md. H. Kabir, and S. C. Paul, "Simulation Based Structural Analysis of an Oil Tanker Bulkhead," *Procedia Engineering*, vol. 194, pp. 517–521, Jan. 2017, https://doi.org/10.1016/j.proeng.2017.08.179.

[5] T. T. Tanny, N. Akter, and O. Md. Amin, "Finite element analysis of container ship's cargo hold using ANSYS and POSEIDON software," *AIP Conference Proceedings*, vol. 1919, no. 1, Dec. 2017, Art. no. 020012, https://doi.org/10.1063/1.5018530.

[6] A. I. Pritykin, "Load Capacity of the Board Grillage with Two Stringers Under Ice Loading," *Marine Intellectual Technologies*, vol. 4, pp. 15–21, 2021, https://doi.org/10.37220/MIT.2021.54.4.001.

[7] D. T. Hang, N. X. Tung, D. V. Tu, and N. N. Lam, "Finite Element Analysis of a Double Beam connected with Elastic Springs," *Engineering, Technology & Applied Science Research*, vol. 14, no. 1, pp. 12482–12487, Feb. 2024, https://doi.org/10.48084/etasr.6489.

[8] D. K. Kim, S. Li, K. Yoo, K. Danasakaran, and N.-K. Cho, "An empirical formula to assess ultimate strength of initially deflected plate: Part 2 = combined longitudinal compression and lateral pressure," *Ocean Engineering*, vol. 252, May 2022, Art. no. 111112, https://doi.org/10.1016/j.oceaneng.2022.111112.

[9] K. N. Anyfantis, "Ultimate strength of stiffened panels subjected to non-uniform thrust," *International Journal of Naval Architecture and Ocean Engineering*, vol. 12, pp. 325–342, Jan. 2020, https://doi.org/10.1016/j.ijnaoe.2020.03.003.

[10] S. Matsui, "Practical estimation method for extreme value distribution of von Mises stress in ship structure," *Journal of Marine Science and Technology*, vol. 28, no. 1, pp. 72–86, Mar. 2023, https://doi.org/10.1007/s00773-022-00904-7.

[11] A. Silva-Campillo, L. Ulla-Campos, J. C. Suárez-Bermejo, and M. A. Herreros-Sierra, "Effect of bow hull form on the buckling strength assessment of the corner bracket connection," *Ocean Engineering*, vol. 265, Dec. 2022, Art. no. 112562, https://doi.org/10.1016/j.oceaneng.2022.112562.

[12] B. Achour, D. Ouinas, M. Touahmia, and M. Boukendakdji, "Buckling of Hybrid Composite Carbon/Epoxy/Aluminum Plates with Cutouts," *Engineering, Technology & Applied Science Research*, vol. 8, no. 1, pp. 2393–2398, Feb. 2018, https://doi.org/10.48084/etasr.1224.

[13] M. Faisal *et al.*, "Rapid hull collapse strength calculations of double hull oil tankers after collisions," *Ships and Offshore Structures*, vol. 12, no. 5, pp. 624–639, Jul. 2017, https://doi.org/10.1080/17445302.2016.1192747.

[14] A. Kuznecovs, M. Schreuder, and J. W. Ringsberg, "Methodology for the simulation of a ship's damage stability and ultimate strength conditions following a collision," *Marine Structures*, vol. 79, Sep. 2021, Art. no. 103027, https://doi.org/10.1016/j.marstruc.2021.103027.

[15] A. Alzahougi, B. Demir, and M. Elitas, "An Optimization Study on Resistance Spot Welding of DP600 Sheet Steel via Experiment and Statistical Analysis," *Engineering, Technology & Applied Science Research*, vol. 13, no. 4, pp. 11106–11111, Aug. 2023, https://doi.org/10.48084/etasr.5804.

[16] S. M. Nguyen, V. L. Phan, N. L. Tran, X. H. Nguyen, and T. H. Nguyen, "Time-Dependent Reliability Assessment of a Continuous I-shaped Steel Beam Considering Corrosion Effects," *Engineering, Technology & Applied Science Research*, vol. 12, no. 6, pp. 9523–9526, Dec. 2022, https://doi.org/10.48084/etasr.5273.

[17] O. Ozguc, "Oil Tanker Simplified Fatigue Assessment with Inspection and Repair Approach and Parameters," *Transactions on Maritime Science*, vol. 10, no. 01, pp. 42–55, Apr. 2021, https://doi.org/10.7225/toms.v10.n01.003.

[18] C. Gonçalves, R. F. Martins, R. Almeida, and L. L. das Neves, "Static and Fatigue Behaviour of the Main Section of a Fast Patrol Boat," *Procedia Engineering*, vol. 74, pp. 161–164, Jan. 2014, https://doi.org/10.1016/j.proeng.2014.06.242.

[19] A. Mansourn and D. Liu, *Strength of Ships and Ocean Structures 2008.* Jersey City, NJ, USA: Society of Naval Architects & Marine Engineers, 2008.

[20] P. Caridis, *Local Strength of Ship Structures.* Boca Raton, FL, USA: CRC Press, 2024.