Journal Applied Science and Technology on Naval Engineering
e-ISSN: 3031-8866

Structural Analysis and Bow Construction Improvement of KMP Portlink III under Collision Load Using the Finite Element Method

Yeremia Parasian Christover Manurung[1,*], Wiwin Sulistyawati[1]

[1]Department of Naval Architecture, Faculty of Engineering, Universitas Pembangunan Nasional "Veteran" Jakarta, Depok 16515, Indonesia
*Corresponding author: yeremiam2@gmail.com

Received: 13/07/2026; Revised: 07/08/2026; Accepted: 07/08/2026; Published: 14/08/2026

Vol. 4, No. 2, August 2026, pp. 30-38
DOI: https://doi.org/10.54378/astne.v4i2v2.04

Abstract

Ship collision accidents can cause severe damage to the bow structure, as it is the first part of the vessel to receive impact loads. This study aims to analyze the structural response of the bow of KMP Portlink III under collision loads and to evaluate the effectiveness of adding carlings in improving structural strength. The bow structure was modeled using Autodesk Fusion, while the numerical analysis was carried out using Autodesk Inventor based on the Finite Element Method (FEM). Four design configurations, namely the original design, horizontal carling, vertical carling, and combined carling, were evaluated under collision loads of 100%, 104%, 106%, 108%, and 110% of the initial impact load of 1,191,900 N. The evaluated parameters were the maximum Von Mises stress and safety factor. The simulation results showed that the original bow structure reached a maximum Von Mises stress of 235–258 MPa with a safety factor of 1.00–0.91, indicating that the structure exceeded the material yield strength under collision loads above the design load. In contrast, the combined carling configuration produced the best structural performance by reducing the maximum Von Mises stress to 78–86 MPa and increasing the safety factor to 3.01–2.74 under all loading conditions. These findings demonstrate that the combined carling reinforcement is the most effective configuration for improving the collision resistance and structural reliability of the bow of KMP Portlink III.

Keywords: Collision load; Finite Element Method; Von Mises Stress; Safety Factor; Carling.

1. Introduction

Ship collisions remain one of the major causes of structural damage in maritime transportation. During a collision, the bow structure receives the initial impact load and experiences the highest stress concentration. Excessive collision loads may cause structural deformation, material yielding, and even structural failure, which can reduce ship safety and increase repair costs. Therefore, evaluating the structural performance of ship bows under collision loading plays an important role in improving structural reliability and operational safety.

Researchers have extensively applied the Finite Element Method (FEM) to investigate the structural response of ships subjected to collision loads. AbuBakar and Dow (2019) analyzed the impact characteristics of ship bows during collisions, while Liu et al. (2018) reviewed experimental and numerical methods for predicting collision damage. Hanif et al. (2023) demonstrated that FEM effectively evaluates the structural strength of stiffened ship panels under various loading conditions. Furthermore, Zhang et al. (2019) validated numerical collision simulations using experimental observations, whereas Xie et al. (2020) showed that reinforcement arrangements significantly influence stress distribution and deformation behavior under impact loading. These studies confirm that FEM provides reliable predictions of stress distribution and structural behavior during collision events.

Despite these developments, previous studies mainly focused on evaluating collision responses without investigating structural reinforcement alternatives for improving bow performance. In addition, limited studies have compared the effectiveness of different carling configurations under increasing collision loads. Consequently, designers still lack quantitative information regarding the influence of horizontal, vertical, and combined carling arrangements on structural performance. This limitation highlights the need for further investigation into reinforcement configurations that can improve collision resistance.

This study addresses this research gap by comparing three carling reinforcement configurations, namely horizontal, vertical, and combined carlings, with the original bow structure of KMP Portlink III. Unlike previous studies that

e-ISSN: 3031-8866

primarily evaluated structural responses, this study focuses on identifying the most effective reinforcement configuration through a comparative numerical analysis. The study evaluates Von Mises stress and safety factor under collision loads ranging from 100% to 110% of the initial collision load using the Finite Element Method. The findings provide structural design recommendations for improving bow collision resistance and offer a practical reference for future reinforcement design and structural assessment of ship bow structures.

The remainder of this paper is organized as follows. Section 2 describes the research methodology, including the numerical model, material properties, finite element modeling, boundary conditions, collision loading, and evaluation procedure. Section 3 presents and discusses the simulation results, including the mesh convergence analysis, the structural response of the original bow, and the effectiveness of the proposed reinforcement configurations. Finally, Section 4 concludes the study by summarizing the main findings, discussing the limitations of the methodology, and providing recommendations for future research.

2. Methodology

2.1 Numerical Model

This study investigated the structural response of the bow of KMP Portlink III under collision loading using the Finite Element Method (FEM). The bow geometry was developed based on the General Arrangement and Profile Construction drawings provided by PT ASDP Indonesia Ferry (Persero). The principal dimensions of the vessel are presented in Table 1.

The three-dimensional model of the bow structure was created using Autodesk Fusion and subsequently imported into Autodesk Inventor Stress Analysis for numerical simulation. The modeled structure included the bow shell plating, transverse frames, longitudinal stiffeners, deck structures, and other supporting members located within the analyzed region. Four structural configurations were investigated, namely the original structure, horizontal carling reinforcement, vertical carling reinforcement, and combined horizontal-vertical carling reinforcement.

2.2 Material Properties and Finite Element Modeling

The numerical simulation was performed using the static linear analysis module available in Autodesk Inventor. The structural material was defined as Steel KI-A, representing normal-strength shipbuilding steel. The material properties used in the analysis consisted of a Young's modulus of 210 GPa, a yield strength of 235 MPa, an ultimate tensile strength of 450 MPa, a Poisson's ratio of 0.30, a density of 7850 kg/m^3, and a shear modulus of 80 GPa.

The finite element model was discretized using automatic tetrahedral elements. To ensure numerical accuracy, a mesh convergence study was conducted using element sizes of 140, 130, 120, 110, and 100 mm. Based on the convergence results, the 100 mm element size was selected because it produced stable Von Mises stress results with 1,944,129 nodes.

2.3 Boundary Conditions and Collision Loading

Boundary conditions were defined by applying fixed supports at the aft end of the bow model, representing the structural continuity between the analyzed bow section and the remaining ship hull. These constraints prevented both translational and rotational displacements during the simulation.

The collision load applied to the bow structure was determined from the ship collision analysis. The kinetic energy associated with the collision can be expressed as

$$[E_k = \frac{1}{2} m v^2] \quad (1)$$

where (E_k) is the collision kinetic energy (J), (m) is the ship mass (kg), and (v) is the collision velocity (m/s).

Based on the collision load calculation, the initial impact force applied to the bow structure was 1,191,900 N. To evaluate the structural performance under increasing loading conditions, five loading cases were considered, corresponding to 100%, 104%, 106%, 108%, and 110% of the initial collision load. The collision load was applied normal to the bow plating to simulate the impact between the vessel and a rigid berth.

2.4 Evaluation Procedure

The structural performance of each reinforcement configuration was evaluated under identical material properties, boundary conditions, mesh parameters, and loading conditions to ensure a consistent comparison. The analysis focused on the maximum Von Mises stress and safety factor obtained from the finite element simulation.

The numerical results were validated using the minimum yield strength requirement specified in the Rules for Hull of Biro Klasifikasi Indonesia (BKI), where normal-strength shipbuilding steel has a minimum yield strength of 235 MPa. Structures producing Von Mises stresses below this value were considered to remain within the elastic region, whereas stresses exceeding the yield strength indicated the onset of plastic deformation. The reinforcement configuration providing the lowest Von Mises stress and the highest safety factor under all loading conditions was identified as the most effective design for improving the collision resistance of the bow structure.

3. Results and Discussion

3.1 Mesh Convergence Analysis

A mesh convergence study was performed to determine the optimum mesh size for the finite element analysis. Six mesh sizes, namely 140, 130, 120, 110, 105, and 100 mm, were evaluated by comparing the maximum Von Mises stress obtained from each simulation. The mesh convergence results are presented in Table 3, while the relationship between mesh size and the maximum Von Mises stress is illustrated in Figure 1.

The results show that the equivalent stress values exhibit slight fluctuations during mesh refinement. However, the results obtained using the 105 mm and 100 mm mesh sizes are relatively close, indicating that the solution begins to stabilize. Therefore, the 100 mm mesh was selected for all subsequent analyses because it provides the highest mesh density and a conservative stress prediction while maintaining sufficient numerical accuracy, with 1,944,129 nodes.

Figure 1 Meshing

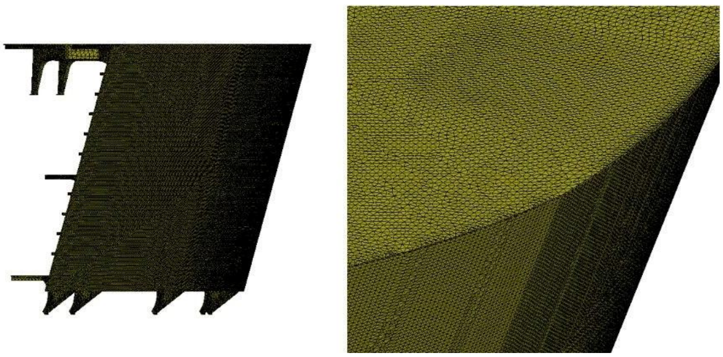


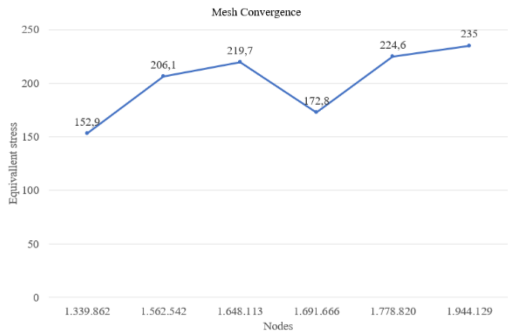
Table 3. Mesh convergence results

Mesh Size (mm)	Nodes	Equivalent Stress (MPa)
140	1.339.862	152
130	1.562.542	206,1
120	1.648.113	219,7
110	1.691.666	172,8
105	1.778.820	224,6

e-ISSN: 3031-8866

100	1.944.129	235

Figure 2. Mesh convergence analysis



3.2 Structural Response of the Original Bow

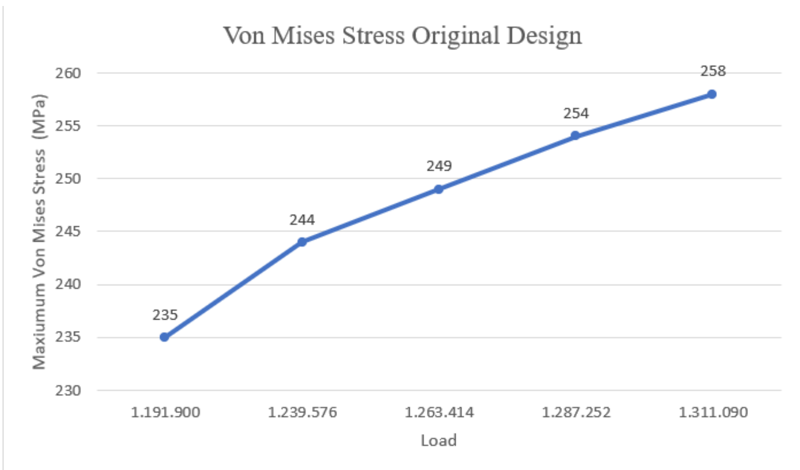
The structural response of the original bow configuration under collision loading is summarized in Table 4, while the Von Mises stress distribution is presented in Figure 2.

The simulation results show that the maximum Von Mises stress increased with increasing collision load. At the design load (100%), the maximum Von Mises stress reached approximately 235 MPa, which remained close to the yield strength of Steel KI-A. As the collision load increased to 104%, 106%, 108%, and 110%, the maximum Von Mises stress exceeded the material yield strength, indicating that the structure had entered the plastic deformation region.

In contrast, the safety factor continuously decreased as the collision load increased. The reduction in the safety factor indicates that the structural capacity of the original bow became insufficient under higher collision loads. These results demonstrate that the original bow structure has limited resistance to collision loads exceeding the design condition.

Table 4. Structural response of the original bow under different collision loads

Load Variation	Load	Maximum von mises stress (MPa)	Safety Factor	Status
100%	1.191.900	235	1	Safe
104%	1.239.576	244	0,96	Not Safe
106%	1.263.414	249	0,94	Not Safe
108%	1.287.252	254	0,93	Not Safe
110%	1.311.090	258	0,91	Not Safe



e-ISSN: 3031-8866

Figure 3. Von Mises stress distribution of the original bow structure

3.3 Effect of Reinforcement Configurations

The influence of the reinforcement configurations on the structural performance of the bow was evaluated by comparing the maximum Von Mises stress and safety factor obtained from each numerical simulation. The comparison results are presented in Table 5, whereas the stress distributions for each reinforcement configuration are shown in Figures 3–5.

The addition of horizontal carlings significantly reduced the maximum Von Mises stress compared with the original structure. The additional stiffeners increased the structural stiffness and distributed the collision load more evenly, thereby reducing stress concentration around the impact area.

The vertical carling configuration also improved the structural performance by increasing the load-carrying capacity of the bow structure. However, the stress reduction achieved by the vertical reinforcement was smaller than that obtained using the horizontal reinforcement.

Among all reinforcement configurations, the combined horizontal and vertical carlings produced the best structural performance. This configuration provided the lowest maximum Von Mises stress and the highest safety factor under all loading conditions. The combination of longitudinal and transverse reinforcement improved the overall structural rigidity, allowing the collision load to be distributed more uniformly throughout the bow structure.

Table 5. Comparison of structural performance for all reinforcement configurations

Design Configuration	Load Variation (%)	Load (N)	Maximum Von Mises Stress (MPa)	Safety Factor	Status
Original Design	100	1.191.900	235	1	Safe
	104	1.239.576	244	0,96	Not Safe
	106	1.263.414	249	0,94	Not Safe
	108	1.287.252	254	0,93	Not Safe
	110	1.311.090	258	0,91	Not Safe
Horizontal Carling	100	1.191.900	137	1,72	Safe
	104	1.239.576	142	1,65	Safe
	106	1.263.414	145	1,62	Safe
	108	1.287.252	148	1,59	Safe
	110	1.311.090	150	1,56	Safe
Vertical Carling	100	1.191.900	147	1,60	Safe
	104	1.239.576	153	1,54	Safe
	106	1.263.414	156	1,51	Safe
	108	1.287.252	159	1,48	Safe
	110	1.311.090	162	1,45	Safe
Combined Carling	100	1.191.900	78	3,01	Safe
	104	1.239.576	81	2,90	Safe
	106	1.263.414	83	2,84	Safe
	108	1.287.252	84	2,79	Safe

	110	1.311.090	86	2,74	Safe

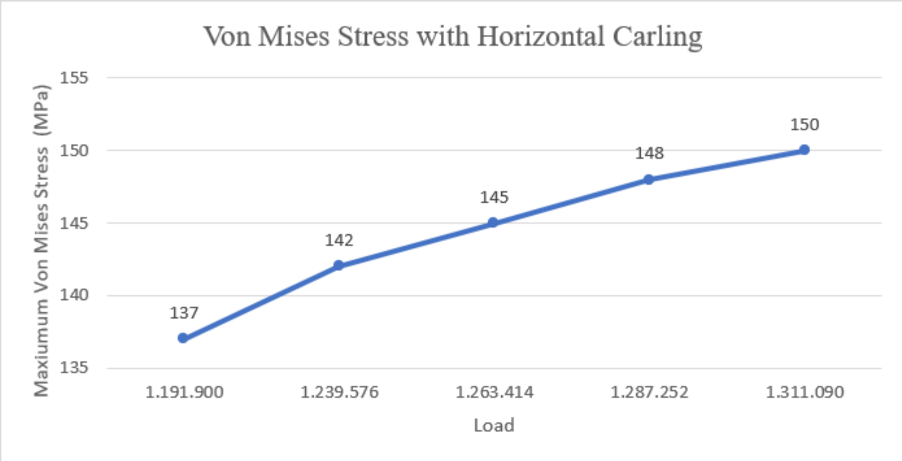


Figure 4. Von Mises stress distribution for the horizontal carling configuration

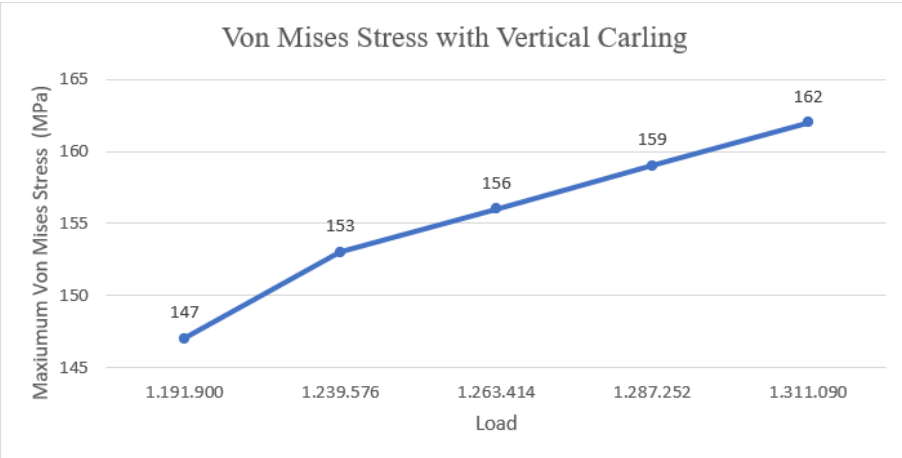


Figure 5. Von Mises stress distribution for the vertical carling configuration

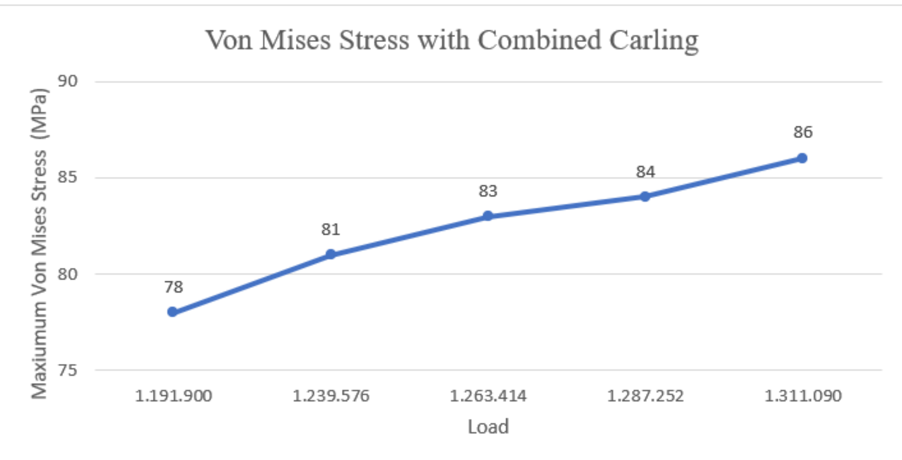


Figure 6. Von Mises stress distribution for the combined carling configuration

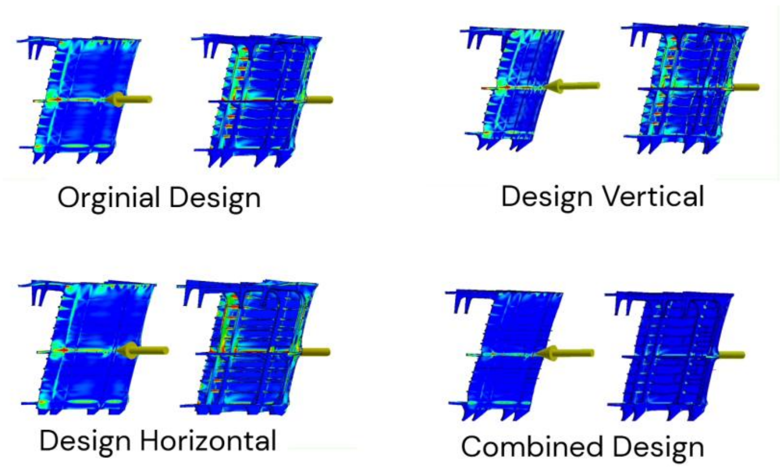


Figure 7. Finite Element Analysis (FEA) results of the bow structure under collision loading for different carling configurations.

3.4 Discussion

The numerical results demonstrate that structural reinforcement effectively improves the collision resistance of the bow structure. The reduction in maximum Von Mises stress and the corresponding increase in the safety factor indicate that the additional carlings enhance the structural stiffness and reduce stress concentration in the impact region.

These findings are consistent with previous studies, which reported that the addition of stiffening members improves the load distribution and structural strength of ship structures subjected to collision loading. However, unlike previous studies that generally evaluated a single reinforcement configuration, the present study compares three different carling arrangements under identical loading conditions. The results demonstrate that the combined horizontal and vertical carling configuration provides the most effective reinforcement for improving the structural performance of the bow of KMP Portlink III.

3.5 Limitations in Methodology

This study has several methodological limitations. First, the finite element analysis was conducted using a linear static approach that does not consider material plasticity, nonlinear contact behavior, or strain-rate effects during collision events. Second, the collision load was represented as an equivalent static load instead of a transient dynamic impact, which may not fully capture the actual structural response during ship collisions. Third, the numerical investigation was limited to the bow structure of KMP Portlink III under a single collision scenario with predefined loading variations ranging from 100% to 110% of the calculated collision load. Therefore, the results should be interpreted as a comparative evaluation of the proposed reinforcement configurations rather than a complete representation of real collision events.

3.6 Recommendations for Future Research

Future research is recommended to incorporate nonlinear finite element analysis with material plasticity and contact interaction to better represent actual collision behavior. Dynamic impact simulations involving different collision velocities, impact angles, and striking ship characteristics should also be considered to improve the accuracy of structural response predictions. Furthermore, optimization of reinforcement configurations with consideration of structural weight and construction feasibility is recommended to obtain more efficient bow designs. Experimental validation or full-scale measurement is also encouraged to verify the numerical results obtained in this study.

4. Conclusion

This study investigated the structural response of the bow of KMP Portlink III under collision loading using the Finite Element Method (FEM) and evaluated the effectiveness of three carling reinforcement configurations. The simulation results showed that increasing the collision load increased the maximum Von Mises stress and reduced the safety factor of the original bow structure. The original design satisfied the safety criterion only under the design collision load (100%), while higher loading conditions caused the stress to exceed the material yield strength.

The addition of carling reinforcements significantly improved the structural performance by reducing stress concentration and increasing the safety factor. Among the evaluated configurations, the combined horizontal–vertical carling arrangement provided the best performance, reducing the maximum Von Mises stress to 78–86 MPa while increasing the safety factor to 3.01–2.74 under all loading conditions. These findings demonstrate that the combined reinforcement configuration is the most effective solution for improving the collision resistance of the bow structure.

Overall, this study provides a practical reference for the structural reinforcement design of ship bows subjected to collision loads and demonstrates the applicability of the Finite Element Method as an effective tool for evaluating alternative reinforcement configurations in ship structures.

Acknowledgments

The authors would like to express their sincere gratitude to the Department of Naval Architecture, Universitas Pembangunan Nasional Veteran Jakarta, for providing academic support and research facilities. The authors also gratefully acknowledge PT ASDP Indonesia Ferry (Persero) for providing the ship data and technical information used in this study. In addition, the authors sincerely thank the supervisors for their valuable guidance, constructive comments, and continuous support throughout this research.

References

AbuBakar, A. and Dow, R.S. (2019) 'The impact analysis characteristics of a ship's bow during collisions', Engineering Failure Analysis, 100, pp. 492–511.

Aditya, M.R., Zakki, A.F. and Adietya, B.A. (2025) 'Analisis tubrukan lateral pada lambung kapal perintis dengan metode elemen hingga', Jurnal Teknik Perkapalan, 13(3).

Alie, M.Z.M. (2016) 'Residual strength analysis of asymmetrically damaged ship hull girder using beam finite element method', Makara Journal of Technology, 20(1), pp. 2–8.

Biro Klasifikasi Indonesia (2022) Rules for Hull, Part 1: Seagoing Ships, Volume II. Jakarta: Biro

Klasifikasi Indonesia. Biro Klasifikasi Indonesia (2025) Rules for Hull, Part 1: Seagoing Ships. Jakarta:

Biro Klasifikasi Indonesia.

Biro Klasifikasi Indonesia (2026) Rules for Hull, Part 1: Seagoing Ships (Consolidated Edition). Jakarta: Biro Klasifikasi Indonesia.

Crudu, L., Neculet, O. and Marcu, O. (2018) 'Hull deflection in still water and in waves of a pipe layer barge', IOP Conference Series: Materials Science and Engineering, 444(6), p. 062002.

Dzikron, N.A. and Yulianto, T. (2019) Analisis Tegangan Haluan Kapal Akibat Tubrukan. Doctoral dissertation. Institut Teknologi Sepuluh Nopember.

Hanif, M.I., Adiputra, R., Prabowo, A.R., Yamada, Y. and Firdaus, N. (2023) 'Assessment of the ultimate strength of stiffened panels of ships considering uncertainties in geometrical aspects: Finite element approach and simplified formula', Ocean Engineering, 286, p. 115522.

Hutagalung, C.I., Bahatmaka, A., Aryadi, W., Putrananda, M.B.S. and Puteri, B.A. (2025) 'Structural analysis of semi-submarine monohull ship for tourism applications by using finite element method', Jurnal Rekayasa Mesin, 16(1), pp. 299–314.

Kawashima, F., Tatsumi, A., Iijima, K., Shiomitsu, D., Yanagimoto, F. and Ishibashi, K. (2024) 'Initial deflection measurement of continuous plates in actual ships using 3D laser scanner', ClassNK Technical Journal, 2024(2), pp. 63–70.

Liu, B., Pedersen, P.T., Zhu, L. and Zhang, S. (2018) 'Review of experiments and calculation procedures for ship collision and grounding damage', Marine Structures, 59, pp. 105–121.

Liu, B., Villavicencio, R., Zhang, S. and Soares, C.G. (2017) 'Assessment of external dynamics and internal mechanics in ship collisions', Ocean Engineering, 141, pp. 326–336.

Xie, H., Liu, F., Tang, H. and Liu, X. (2020) 'Numerical study on the dynamic response of a truncated ship-hull structure under asymmetrical slamming', Marine Structures, 72, p. 102767.

Yu, K., Wang, H., Liu, X. and Peng, B. (2023) 'Research on a ship deflection anti-collision method based on a water-jet interference flow field', Applied Sciences, 13(13), p. 7354.

Zhang, S., Villavicencio, R., Zhu, L. and Pedersen, P.T. (2019) 'Ship collision damage assessment and validation with experiments and numerical simulations', Marine Structures, 63, pp. 239–256.