

Special Issue Reprint

Advanced Analysis of Marine Structures

Edition II

Edited by
Bin Liu, Chenfeng Li and Kun Liu

mdpi.com/journal/jmse

Advanced Analysis of Marine Structures—Edition II

Advanced Analysis of Marine Structures—Edition II

Guest Editors

Bin Liu

Chenfeng Li

Kun Liu



Basel • Beijing • Wuhan • Barcelona • Belgrade • Novi Sad • Cluj • Manchester

Guest Editors

Bin Liu
Green & Smart
River-Sea-Going Ship, Cruise
and Yacht Research Centre
Wuhan University of
Technology
Wuhan
China

Chenfeng Li
College of Shipbuilding
Engineering
Harbin Engineering
University
Harbin
China

Kun Liu
School of Naval Architecture
and Ocean Engineering
Jiangsu University of Science
and Technology
Zhenjiang
China

Editorial Office

MDPI AG
Grosspeteranlage 5
4052 Basel, Switzerland

This is a reprint of the Special Issue, published open access by the journal *Journal of Marine Science and Engineering* (ISSN 2077-1312), freely accessible at: https://www.mdpi.com/journal/jmse/special_issues/QT5O4B009Q.

For citation purposes, cite each article independently as indicated on the article page online and as indicated below:

Lastname, A.A.; Lastname, B.B. Article Title. <i>Journal Name</i> Year , Volume Number, Page Range.
--

ISBN 978-3-7258-7538-2 (Hbk)

ISBN 978-3-7258-7539-9 (PDF)

<https://doi.org/10.3390/books978-3-7258-7539-9>

© 2026 by the authors. Articles in this reprint are Open Access and distributed under the Creative Commons Attribution (CC BY) license. The reprint as a whole is distributed by MDPI under the terms and conditions of the Creative Commons Attribution-NonCommercial-NoDerivs (CC BY-NC-ND) license (<https://creativecommons.org/licenses/by-nc-nd/4.0/>).

Contents

Bin Liu, Chenfeng Kun Liu

Advanced Analysis of Marine Structures—Edition II

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 1763, <https://doi.org/10.3390/jmse12101763> 1

Shiye Liu, Kun Liu, Hewei Liu, Shuai Zong, Yue Lu and Chuhao Liu

Calibration of the Modified Mohr–Coulomb Failure Criterion and Its Application in the Study of Collision Response of Ship Hull Plate Frame Structures

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 805, <https://doi.org/10.3390/jmse12050805> 4

Jinsong Xia, Tongqiang Yu, Kun Liu, Junji Xiang, Nan Zhao and Junjie Liu

Experimental Investigation of Ice Loads on Structures during Quasi-Static Compression Tests

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 924, <https://doi.org/10.3390/jmse12060924> 28

Tao Lu, Jiaxia Wang, Kun Liu and Xiaochao Zhao

Experimental and Numerical Prediction of Slamming Impact Loads Considering Fluid–Structure Interactions

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 733, <https://doi.org/10.3390/jmse12050733> 43

Weilong Zhang, Jinju Cui and Deyu Wang

A Two-Step Approach for Evaluating the Dynamic Ultimate Load Capacity of Ship Structures

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 219, <https://doi.org/10.3390/jmse12020219> 67

Chen Chen, Hong Zhou, Zhengda Lv and Ziqiu Li

Experimental and Numerical Studies on the Ultimate Bending Strength of Welded Plated Grillage with Combined Openings

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 295, <https://doi.org/10.3390/jmse12020295> 91

Lei Ao, Ziqi Ding, Bin Liu, Zhiyong Pei, Qin Tang and Weiguo Wu

Experimental and Numerical Analysis of Ultimate Carrying Capacity of a Funnel Structure with Opening under Wind Pressure

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 41, <https://doi.org/10.3390/jmse12010041> 112

Zhixin Xiong, Chengyuan Zhu, Yue Yang, Tong Lin and Ruoxuan Li

XFEM-Based Study of Fatigue Crack Propagation in Rocket Deflector Troughs under Coupled High-Temperature and Impact Conditions

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 207, <https://doi.org/10.3390/jmse12020207> 134

Mengxiang Li, Guo Wang, Kun Liu, Yue Lu and Jiaxia Wang

Experimental and Numerical Analysis of Supporting Forces and Lashing Forces in a Ship Cargo Securing Scheme

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 158, <https://doi.org/10.3390/jmse12010158> 150

Zhiqiang Li, Qiushi Liu, Shian Liu, Xueying Liu, Yongqiang Zhang, Shixiang Jia, et al.

Dynamic Stability Analysis of Subsea Tunnel Crossing Active Fault Zone: A Case Study

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 839, <https://doi.org/10.3390/jmse12050839> 173

Ruili Guo, Hongyang Hu, Haisheng Zhao and Yao Zhang

Stress Concentration Factors Due to Misalignment at Girth Welds in Bi-Layer Pipes

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 231, <https://doi.org/10.3390/jmse12020231> 186

Qiang Wang, Lihong Wu, Chenfeng Li, Xin Chang and Boran Zhang

Research on a Real-Time Prediction Method of Hull Girder Loads Based on Different Recurrent Neural Network Models

Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 746, <https://doi.org/10.3390/jmse12050746> 204

- Pengyu Wei, Chunlong Li, Ze Jiang and Deyu Wang**
Real-Time Digital Twin of Ship Structure Deformation Field Based on the Inverse Finite Element Method
Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 257, <https://doi.org/10.3390/jmse12020257> **225**
- Jian Leng, Libin Liu, Chencong Liao and Guanlin Ye**
Fully Buried Pipeline Flootation in Poro-Elastoplastic Seabed under Combined Wave and Current Loadings
Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 995, <https://doi.org/10.3390/jmse12060995> **243**
- Tongzheng Zhang, Junbo Zhang, Qinglian Hou, Gangyi Yu, Ao Chen, Zhiqiang Liu and Rong Wan**
Numerical Modelling of the Hydrodynamic Performance of Biodegradable Drifting Fish Aggregating Devices in Currents
Reprinted from: *J. Mar. Sci. Eng.* **2024**, 12, 1059, <https://doi.org/10.3390/jmse12071059> **260**

Advanced Analysis of Marine Structures—Edition II

Bin Liu ¹, Chenfeng Li ² and Kun Liu ^{3,*}

¹ Green & Smart River-Sea-Going Ship, Cruise and Yacht Research Centre, Wuhan University of Technology, Wuhan 430063, China; liubin8502@whut.edu.cn

² College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China; lichenfeng@hrbeu.edu.cn

³ School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang 212100, China

* Correspondence: kunliu@just.edu.cn

One of the key issues in the design of modern ship and offshore structures is the accurate prediction of strength under various load conditions, especially impact, ultimate and fatigue strength. The advanced analysis of marine structures is involved in the mechanical analyses of advanced materials such as alloys and composite materials, and strength analyses of novel structures such as sandwich structures, in order to render marine structures lightweight, safe and economical throughout their lifetimes. Accordingly, the Special Issue “Advanced Analysis of Marine Structures” was prepared to collect works relating to advanced analysis of marine structures in general. To continue that work, the present Special Issue, “Advanced Analysis of Marine Structures—Edition II”, was developed to include more works related to this topic.

The present Special Issue contains 14 articles: 3 papers on the impact response of ship structures, 3 papers related to the ultimate strength of ship structures, 1 paper dealing with the structural fatigue strength, 3 papers covering the various aspects of strength assessment of ship and offshore structures, 2 papers revolving around the structural load prediction and deformation monitoring, and 2 papers involved in the hydrodynamic analysis of marine devices.

The accurate prediction of ship structural damage due to collisions is a research focus in impact strength analysis. To enhance the predictive accuracy of failure criteria in the simulation of ship collisions, Liu et al. [1] studied the modified Mohr–Coulomb failure criterion for metals, utilizing experimental and numerical methods for parameter calibration.

The design of polar ship structures is often based on the rules and regulations of classification societies, but ice loading tends to be conservative and exhibits redundancy. Therefore, Xia et al. [2] evaluated the applicability of the empirical formula for ice load assessments by conducting a series of quasi-static indentation tests on scaled hull plates under laboratory-made ice blocks of different scales.

Slamming impact is a typical load on a ship bow, and the whipping induced by slamming can significantly increase ship loading. Thus, Lu et al. [3] carried out an experimental and numerical study on the water entry of rigid wedges with various deadrise angles. The drop height and deadrise angle were parametrically varied to investigate the effect of the entry velocity and wedge shape on the impact dynamics.

As the dynamic ultimate strength of ship structures is a hot topic in ship safety assessment, Zhang et al. [4] proposed a novel two-step approach for determining the dynamic ultimate load capacity of ship structures. The main idea behind the two-step approach is to determine the dynamic ultimate load capacity based on the static ultimate load capacity after accounting for impacts that cause strain on the ship structures.

Plated grillage with openings is a typical structural form in the superstructure of large passenger ships, and the deformation and stresses generated during the welding complicate the structural failure behavior. Chen et al. [5], therefore, used a new partitioned

inherent strain method and nonlinear finite element method to simulate the welding and loading process, validated by experiments.

Since funnels have a non-typical structure in large passenger ships, Ao et al. [6] presented experimental and finite element investigations on the ultimate strength of the funnel structure of a large passenger ship subjected to wind pressure.

Offshore platforms can be used to launch rockets, so Xiong et al. [7] investigated fatigue crack propagation on the lower surface of rocket deflector troughs in offshore rocket launch platforms. They analyzed the fatigue crack propagation patterns in both the maximum stress and maximum deformation regions of the deflector channels under the combined conditions of high temperature and impact.

Safety assessments of ship cargo-securing systems is of significant importance in preventing casualties, vessel instability and economic losses. Li et al. [8], thus, adopted an independently designed cylindrical cargo securing scheme with supporting structures for investigation.

A subsea tunnel crossing an active fault zone is extremely susceptible to disasters, such as tunnel collapses, sudden water ingress and mud inrush. Therefore, Li et al. [9] conducted numerical simulations by employing different lining stiffnesses for tunnel excavation, as well as applying dynamic loads. The dynamic stability of a subsea tunnel crossing an active fault zone was thus evaluated by comparing and analyzing the lining's displacement, peak acceleration and stress characteristics.

Bi-layer pipes, composed of an inner layer and an outer layer, have been widely used in offshore engineering. Guo et al. [10] derived the governing equation for a bi-layer pipe subjected to axisymmetric loadings, and the equation is used to develop stress concentration factor formulations for girth welds in bi-layer pipes.

As the real-time prediction of hull girder loads is of great significance for the safety of ship structures, Wang et al. [11] compared four recurrent neural network models and analyzed the effect of different levels of noise on the prediction accuracy of various load components.

Digital twins play a seminal role in the digital transformation and intelligent upgrade of ships, so Wei et al. [12] introduced a digital twin methodology for the real-time monitoring of ship structure deformation fields based on finite discrete strain data, and a visualization tool framework was developed using virtual reality technology.

The floatation capacity of seabed pipelines has been considered a key risk element during design. Therefore, Leng et al. [13] presented a two-dimensional coupled approach with an elastoplastic theory to study the floatation of pipelines with the combined loading of waves and currents.

A fish aggregating device is an essential supplementary structure used in fishing. Its hydrodynamic performance affects structural and environmental safety in the harsh marine environment. As such, Zhang et al. [14] examined the hydrodynamic performance of biodegradable drifting fish aggregating devices in oceanic currents using numerical modeling.

In summary, the articles presented in the present Special Issue cover broad research topics related to the advanced analysis of marine structures, guiding readers through the best analysis approach.

Author Contributions: Writing—original draft preparation, B.L.; writing—review and editing, K.L. and C.L. All authors have read and agreed to the published version of the manuscript.

Conflicts of Interest: The authors declare no conflicts of interest.

List of Contributions:

1. Liu, S.; Liu, K.; Liu, H.; Zong, S.; Lu, Y.; Liu, C. Calibration of the Modified Mohr–Coulomb Failure Criterion and Its Application in the Study of Collision Response of Ship Hull Plate Frame Structures. *J. Mar. Sci. Eng.* **2024**, *12*, 805. <https://doi.org/10.3390/jmse12050805>
2. Xia, J.; Yu, T.; Liu, K.; Xiang, J.; Zhao, N.; Liu, J. Experimental Investigation of Ice Loads on Structures during Quasi-Static Compression Tests. *J. Mar. Sci. Eng.* **2024**, *12*, 924. <https://doi.org/10.3390/jmse12060924>

3. Lu, T.; Wang, J.; Liu, K.; Zhao, X. Experimental and Numerical Prediction of Slamming Impact Loads Considering Fluid–Structure Interactions. *J. Mar. Sci. Eng.* **2024**, *12*, 733. <https://doi.org/10.3390/jmse12050733>
4. Zhang, W.; Cui, J.; Wang, D. A Two-Step Approach for Evaluating the Dynamic Ultimate Load Capacity of Ship Structures. *J. Mar. Sci. Eng.* **2024**, *12*, 219. <https://doi.org/10.3390/jmse12020219>
5. Chen, C.; Zhou, H.; Lv, Z.; Li, Z. Experimental and Numerical Studies on the Ultimate Bending Strength of Welded Plated Grillage with Combined Openings. *J. Mar. Sci. Eng.* **2024**, *12*, 295. <https://doi.org/10.3390/jmse12020295>
6. Ao, L.; Ding, Z.; Liu, B.; Pei, Z.; Tang, Q.; Wu, W. Experimental and Numerical Analysis of Ultimate Carrying Capacity of a Funnel Structure with Opening under Wind Pressure. *J. Mar. Sci. Eng.* **2024**, *12*, 41. <https://doi.org/10.3390/jmse12010041>
7. Xiong, Z.; Zhu, C.; Yang, Y.; Lin, T.; Li, R. XFEM-Based Study of Fatigue Crack Propagation in Rocket Deflector Troughs under Coupled High-Temperature and Impact Conditions. *J. Mar. Sci. Eng.* **2024**, *12*, 207. <https://doi.org/10.3390/jmse12020207>
8. Li, M.; Wang, G.; Liu, K.; Lu, Y.; Wang, J. Experimental and Numerical Analysis of Supporting Forces and Lashing Forces in a Ship Cargo Securing Scheme. *J. Mar. Sci. Eng.* **2024**, *12*, 158. <https://doi.org/10.3390/jmse12010158>
9. Li, Z.; Liu, Q.; Liu, S.; Liu, X.; Zhang, Y.; Jia, S.; Song, G.; Zhao, Y. Dynamic Stability Analysis of Subsea Tunnel Crossing Active Fault Zone: A Case Study. *J. Mar. Sci. Eng.* **2024**, *12*, 839. <https://doi.org/10.3390/jmse12050839>
10. Guo, R.; Hu, H.; Zhao, H.; Zhang, Y. Stress Concentration Factors Due to Misalignment at Girth Welds in Bi-Layer Pipes. *J. Mar. Sci. Eng.* **2024**, *12*, 231. <https://doi.org/10.3390/jmse12020231>
11. Wang, Q.; Wu, L.; Li, C.; Chang, X.; Zhang, B. Research on a Real-Time Prediction Method of Hull Girder Loads Based on Different Recurrent Neural Network Models. *J. Mar. Sci. Eng.* **2024**, *12*, 746. <https://doi.org/10.3390/jmse12050746>
12. Wei, P.; Li, C.; Jiang, Z.; Wang, D. Real-Time Digital Twin of Ship Structure Deformation Field Based on the Inverse Finite Element Method. *J. Mar. Sci. Eng.* **2024**, *12*, 257. <https://doi.org/10.3390/jmse12020257>
13. Leng, J.; Liu, L.; Liao, C.; Ye, G. Fully Buried Pipeline Flootation in Poro-Elastoplastic Seabed under Combined Wave and Current Loadings. *J. Mar. Sci. Eng.* **2024**, *12*, 995. <https://doi.org/10.3390/jmse12060995>
14. Zhang, T.; Zhang, J.; Hou, Q.; Yu, G.; Chen, A.; Liu, Z.; Wan, R. Numerical Modelling of the Hydrodynamic Performance of Biodegradable Drifting Fish Aggregating Devices in Currents. *J. Mar. Sci. Eng.* **2024**, *12*, 1059. <https://doi.org/10.3390/jmse12071059>

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Calibration of the Modified Mohr–Coulomb Failure Criterion and Its Application in the Study of Collision Response of Ship Hull Plate Frame Structures

Shiye Liu ¹, Kun Liu ^{1,*}, Hewei Liu ², Shuai Zong ¹, Yue Lu ¹ and Chuhao Liu ¹

¹ School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang 212003, China; l917715625@gmail.com (S.L.); 220110101103@stu.just.cn (S.Z.); yuelu@just.edu.cn (Y.L.); sunflee728@gmail.com (C.L.)

² School of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Wuhan 430074, China; wssls1@hust.edu.cn

* Correspondence: kunliu@just.edu.cn; Tel.: +86-135-1169-2085; Fax: +86-0511-8444-6543

Abstract: Within the lifecycle of a ship's hull structure, damage due to collisions has been a focal point of research for researchers both domestically and internationally. To enhance the predictive accuracy of failure criteria in the simulation of ship hull collisions, this paper focuses on the modified Mohr–Coulomb (MMC) failure criterion for metals, utilizing a hybrid experimental–numerical method for parameter calibration. Consideration of stress-state-dependent mesh size sensitivity has been amended, and the approach is integrated into the comprehensive nonlinear finite element software Abaqus 2020. Finite element tensile simulations were conducted to validate the effectiveness of the MMC criterion. Simulation analyses were conducted based on drop hammer collision experiments with various failure criteria and grid sizes. The comparative validation highlighted the superiority of the mesh size sensitivity-corrected MMC failure criterion. The outcomes of this research provide a foundation for assessing the structural safety of ship hulls.

Keywords: ship collision; MMC failure criterion; numerical simulation

1. Introduction

As computational power rapidly advances and finite element analysis (FEA) software continues to evolve, finite element methods have become an integral tool in the field of ship collision studies. Ships, being thin-walled structures, are generally modeled using shell elements in FEA, and failure of ship structures is often represented by the deletion of failed shell elements [1,2]. Thus, accurate prediction of the hull structural failure mode presupposes the definition of an appropriate failure criterion within the simulation software, which has traditionally been a major challenge in ship collision simulations.

Researchers at home and abroad have conducted extensive research on collision damage to ship metal structures. Minorsky [3] categorized the research on ship issues into two aspects: the internal mechanisms and the external mechanisms. The external mechanisms consider the ship as a rigid body, focusing on the six degrees of freedom of rigid motion, external loads applied to the ship, and the dissipation of kinetic energy. The internal mechanisms, on the other hand, are primarily concerned with the dynamic response of the ship's structure and its damage and deformation during a collision. Haris and Amdahl [4] proposed a procedure utilizing a simplified analysis method for examining ship collisions that takes into account the interaction between the impact deformation and the struck ship. Törnqvist [5] conducted research on the design of collision-resistant structures for ships in which the Rice–Tracey and Cockcroft–Latham (RTCL) failure criterion is calibrated and integrated into the LS-DYNA R12 simulation environment. Ehlers et al. [6] determined the impact of various failure criteria, such as mesh size sensitivity, on the

collision results by simulating the collision response of three distinct ship side structures using the finite element method. Storheim et al. [7] investigated the robustness of the fracture criterion, i.e., the ability to simulate fracture under different stress states and mesh sizes.

Concerning metal failure criteria, researchers have conducted numerous studies. Driemeier et al. [8] found that the introduction of notches in tensile specimens can effectively increase the stress triaxiality at the fracture location, leading to earlier failure of the specimen. Wang Zeping [9] integrated the Johnson–Cook (JC) and Gurson–Tvergaard–Needleman (GTN) models to develop an enhanced material model that addresses the GTN model’s accuracy limitations at high strain rates. In an exhaustive review of failure criteria employed in finite element analyses of ship collision events, M.A.G. Calle [10] identified the equivalent plastic strain failure criterion as notably suitable due to its high practicality and straightforward integration into prevalent finite element software.

Recent studies demonstrate that metal fracture strain is not only related to stress triaxiality but also to the Lode angle parameter. Bai and Wierzbicki [11], by modifying the Mohr–Coulomb criterion to predict the ductile fracture of metallic materials, proposed the modified Mohr–Coulomb (MMC). Lou et al. [12] and Hu et al. [13] scrutinized the precision and suitability of the ductile fracture criterion across varying stress triaxiality ranges using AA2024-T351 experimental data as compiled by Bao and Wierzbicki [14]. Their findings disclosed that conventional failure criteria, confined to stress triaxiality considerations, falter in accurately forecasting experimental outcomes across an extended triaxiality spectrum. In contrast, failure criteria that incorporate both stress triaxiality and the Lode angle parameter have demonstrated superior predictive capabilities across a comprehensive array of experimental conditions.

Compared to other failure criteria, the MMC failure criterion takes into account the effects of both stress triaxiality and the Lode angle on failure, and is able to forecast metal failure across a wider range of stress triaxialities and effectively simulate collision damage to metal structures under complex stress conditions. Therefore, it is essential to undertake the calibration of the MMC failure criterion and the mesh-size-based calibration and apply these to the collision response analysis of the ship’s hull panel–frame structure.

The MMC fracture criterion was used by Talebi-Ghadikolaee et al. [15] to examine the fracture behavior of an AA6061-T6 aluminum alloy sheet during U-shaped bending, while Voormeeren et al. [16] talked about how to apply the MMC failure criterion based on a single tensile test. The results of Zhu et al. [17]’s fine mesh simulation demonstrate that the shear force predicted by the MMC failure criterion is essentially in agreement with the data from the experiments.

In this study, to enable a more precise simulation of ship collision problems, a calibration was performed on the MMC failure criterion that accounts for stress triaxiality. The criterion was embedded into the large-scale nonlinear finite element software Abaqus via secondary development. Additionally, drop hammer tests on the hull panel–frame structure using the MMC failure criterion were conducted, along with corresponding simulations to investigate the deformation and damage mechanisms. By discussing the impacts of different failure criteria and mesh sizes on the simulation results, the validation of the MMC failure criterion’s superiority was confirmed following the adjustment for mesh sensitivity. This provides a basis for the safety assessment of hull girder structures.

2. MMC Failure Criterion

The Mohr–Coulomb criterion [18] is a type of stress-based failure criterion that is commonly used to evaluate failure in materials like rock, soil bodies, concrete, etc., that either fail in the elastic stage or only undergo slight plastic deformations. The Mohr–Coulomb criterion is expressed as follows in Equation (1):

$$\max(\tau + c_1\sigma_n)_f = c_2 \quad (1)$$

Here, τ and σ_n represent the shear stress and principal stresses on the fracture plane, respectively. c_2 is the shear strength and c_1 is the friction coefficient, which describes the relative weighting between the shear stress and normal stress. When $c_1 = 0$, the Mohr–Coulomb criterion degenerates into the maximum shear stress criterion. Therefore, the Mohr–Coulomb criterion can be viewed as an extension of the maximum shear stress criterion, as it can simultaneously describe fracture behavior caused by both normal and shear stresses.

Since any stress inside a material can be represented in the Cartesian coordinate system $(\sigma_1, \sigma_2, \sigma_3)$ formed by the three principal stresses, any stress can also be expressed in the cylindrical coordinate system $(\bar{\sigma}, \eta, \bar{\theta})$ formed by the equivalent stress, stress triaxiality, and Lode angle parameters. Bao and Wierzbicki [19] extended the application range of the Mohr–Coulomb criterion by transforming Equation (1) into a function of $(\bar{\sigma}, \eta, \bar{\theta})$ and proposed the MMC failure criterion for predicting ductile fracture in metallic materials. Its original expression is shown in Equation (2):

$$\bar{\epsilon}_f = \left\{ \frac{K}{c_2} [1 - c_\eta(\eta - \eta_0)] \times \left[c_\theta^s + \frac{\sqrt{3}}{2-\sqrt{3}} (c_\theta^{ax} - c_\theta^s) \left(\sec\left(\frac{\bar{\theta}\pi}{6}\right) - 1 \right) \right] \right\}^{-\frac{1}{n}} \times \left[\frac{\sqrt{1+c_1^2}}{3} \cos\left(\frac{\bar{\theta}\pi}{6}\right) + c_1 \left(\eta + \frac{1}{3} \sin\left(\frac{\bar{\theta}\pi}{6}\right) \right) \right] \quad (2)$$

$$c_\theta^{ax} = \begin{cases} 1, & \bar{\theta} \geq 0 \\ c_\theta^c, & \bar{\theta} < 0 \end{cases} \quad (3)$$

where K and n are the hardening coefficients and hardening exponents, respectively, in the hardening criteria, which can be obtained by fitting the true stress–strain curve from uniaxial tension tests. c_1 and c_2 are material parameters in the original Mohr–Coulomb criterion, which can be calibrated through a combination of experimental tests and numerical simulations using optimization algorithms to determine the parameters of the three-dimensional fracture surface, c_1 and c_2 . Parameter c_θ^c, c_θ^s is associated with the asymmetry of the fracture surface. When the material follows the von Mises yield criterion (i.e., isotropic hardening), $c_\eta = 0$ and $c_\theta^s = c_\theta^c = 1$, Equations (2) and (3) can be simplified as follows:

$$\bar{\epsilon}_f = \left\{ \frac{K}{c_2} \left[\frac{\sqrt{1+c_1^2}}{3} \cos\left(\frac{\bar{\theta}\pi}{6}\right) + c_1 \left(\eta + \frac{1}{3} \sin\left(\frac{\bar{\theta}\pi}{6}\right) \right) \right] \right\}^{-\frac{1}{n}} \quad (4)$$

In the plane stress state, there is a one-to-one correspondence between the Lode angle parameter and stress triaxiality, as shown in Equation (5). By substituting Equation (5) into Equation (4), which represents the fracture surface of the MMC failure criterion in $(\bar{\theta}, \eta, \bar{\epsilon}_f)$ three-dimensional (3D) space, the fracture failure trajectory in $(\eta, \bar{\epsilon}_f)$ two-dimensional (2D) space can be obtained and is displayed in Figure 1.

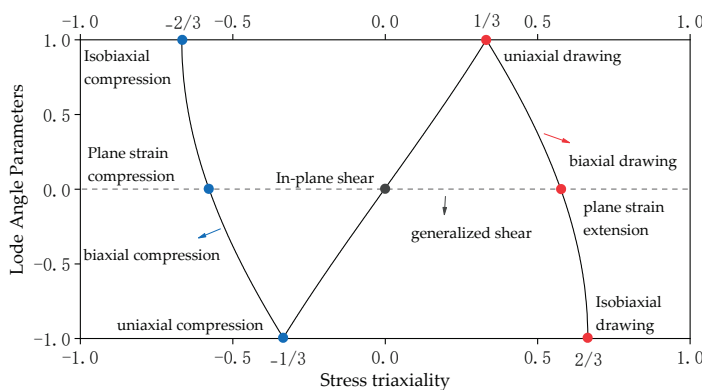


Figure 1. Stress states in the dimension of $(\eta, \bar{\theta})$.

$$\bar{\theta} = -\frac{2}{\pi} \arcsin \left[\frac{27}{2} \eta \left(\eta^2 - \frac{1}{3} \right) \right] \quad (5)$$

2.1. MMC Failure Criterion Parameter Fitting

In the parameter calibration process of the MMC failure criterion, Bai and Wierzbicki [11] adopted the basic procedure proposed for determining the failure criterion parameters in the $(\eta, \bar{\epsilon}_f)$ two-dimensional space, and further applied it to the calibration of parameters for the three-dimensional fracture surface in the $(\eta, \bar{\epsilon}_f, \bar{\theta})$ space. The specific steps are as follows:

Firstly, material testing is performed on specimens under different stress states to obtain their load–displacement curves. According to the setup of the experimental conditions, corresponding finite element simulation simulations are carried out. Critical data, such as the equivalent strain, stress triaxiality, and Lode angle parameters, are extracted from the finite element model, and their evolution curves are plotted with respect to the equivalent strain changes over the course of stress triaxiality and Lode angle parameters. Based on these curves, the average stress triaxiality and average Lode angle or average Lode angle parameters (where this study's MMC fitting selects the average Lode angle parameter) are calculated using Equation (6). The fracture strain, average stress triaxiality, and average Lode angle parameters obtained from these calculations are then used to calibrate the undetermined parameters within the fracture criterion.

$$\begin{cases} \bar{\eta} = \frac{1}{\bar{\epsilon}_f} \int_0^{\bar{\epsilon}_f} \eta(\bar{\epsilon}) d\bar{\epsilon} \\ \bar{L} = \frac{1}{\bar{\epsilon}_f} \int_0^{\bar{\epsilon}_f} L(\bar{\epsilon}) d\bar{\epsilon} \\ \bar{\theta} = \frac{1}{\bar{\epsilon}_f} \int_0^{\bar{\epsilon}_f} \bar{\theta}(\bar{\epsilon}) d\bar{\epsilon} \end{cases} \quad (6)$$

Yu [20] conducted experimental and simulation studies on the plastic and fracture characteristics of marine high-strength steel under different stress states. Their tensile tests primarily included tensile and shear stress states. The literature provides the average stress triaxiality and average Lode angle parameters of the critical element at the failure location of different specimens, along with the corresponding failure strain (see Table 1). This paper, based on the data from the literature, employs MATLAB 2018b optimization tools to fit the MMC failure criterion.

Table 1. Fracture strain under different stress triaxiality and Lode angle.

Sample Type	Specimen Number	Stress Triaxiality η	Lode Angle $\bar{\theta}$	Failure Strain $\bar{\epsilon}_f$
Notched round bar	NRB6	0.443	1	1.091
	NRB1.5	0.716	1	0.910
Grooved plate	FGP12	0.593	0	0.905
	FGP1.8	0.765	0	0.830
Notched plate	NP20	0.446	0.614	1.018
	NP6.67	0.554	0.213	0.973
Shear	SH	−0.037	0.024	0.859

Generally speaking, the structural steel of a ship's hull can be considered an isotropic material. The MMC failure criterion can be simplified from Equations (2)–(4) based on the von Mises criterion's premises, where K and n reference the parameters of the Swift model in the literature [20], which are $K = 909.29$ MPa, $n = 0.1992$. Equation (7) is obtained.

$$\bar{\epsilon}_f = \left\{ \frac{909.29}{c_2} \left[\frac{\sqrt{1+c_1^2}}{3} \cos\left(\frac{\bar{\theta}\pi}{6}\right) + c_1 \left(\eta + \frac{1}{3} \sin\left(\frac{\bar{\theta}\pi}{6}\right) \right) \right] \right\}^{-\frac{1}{0.1992}} \quad (7)$$

At this point, the MMC failure criterion only involves two undetermined parameters, c_1 and c_2 . Utilizing the data from Table 1 and the optimization tool lsqcurvefit in MATLAB, the parameters c_1 and c_2 in Equation (7) are optimized to obtain the best-fitting 3D fracture surface. The lsqcurvefit function in MATLAB requires an initial set of solutions c_1 and c_2 . By substituting the data from specimens 4 and 5 into the aforementioned equation, initial values of $c_1 = -0.0207$ and $c_2 = 278.23$ MPa can be determined. Taking these values as the initial solutions and applying the least squares error optimization, the final MMC failure criterion parameters can be acquired. The MMC failure criterion parameters are then listed in Table 2.

Table 2. Parameters in the MMC failure criterion.

K	N	c_1	c_2
909.29 MPa	0.1992	0.0420	301.47 MPa

The final derived MMC failure criterion is presented as Equation (8):

$$\bar{\epsilon}_f = \left\{ \frac{909.29}{301.47} \left[\frac{\sqrt{1+0.042^2}}{3} \cos\left(\frac{\bar{\theta}\pi}{6}\right) + 0.042 \left(\eta + \frac{1}{3} \sin\left(\frac{\bar{\theta}\pi}{6}\right) \right) \right] \right\}^{-\frac{1}{0.1992}} \quad (8)$$

Using the cftool utility in MATLAB, the aforementioned equation is plotted in the $(\eta, \bar{\theta}, \bar{\epsilon}_f)$ space as shown in Figure 2, where the red line represents the fracture trajectory under the plane stress state.

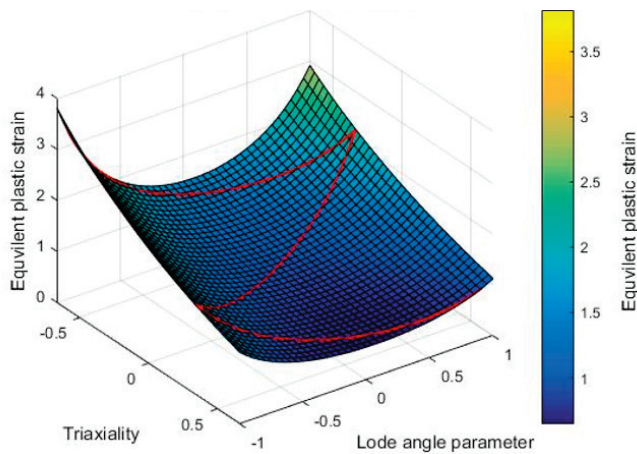


Figure 2. The MMC failure fracture surface of marine high-strength steel.

In ship collision simulation computations, where the finite element type employed is a plane stress shell element, the relationship between the stress triaxiality and the Lode angle parameter under plane stress conditions, as described in Equation (5), is substituted into Equation (8). This yields the fracture trajectory for the MMC failure criterion under plane stress conditions. The curve within the $(\eta, \bar{\epsilon}_f)$ space is depicted in Figure 3.

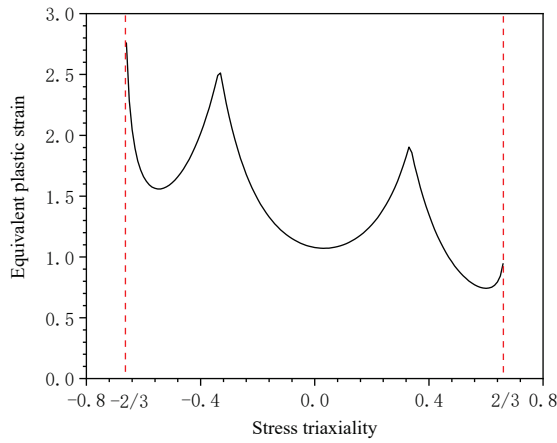


Figure 3. MMC failure locus in $(\eta, \bar{\epsilon}_f)$ dimension.

It should be noted that the parameters for the MMC failure criterion obtained here are based on solid elements with a size of 0.1 mm acquired through experimental–simulation hybrid analysis. These parameters are not directly applicable to ship collision simulations that employ shell elements. Consequently, a modification is necessary to adapt them for use in the simulation analysis of ship structural collisions.

2.2. Mesh Size Sensitivity Correction of the MMC Failure Criterion

The MMC failure criterion is established based on solid elements [11]. When this criterion is applied to shell elements, which operate under the assumption of plane stress and thus neglect variations in stress across the thickness, it results in a heightened sensitivity to the size of the shell element mesh. As the grid size of the shell elements is substantially greater than the plate thickness, in conducting finite element failure simulations with ductile metal plate materials using shell elements, scaling of the failure strain becomes necessary when the dimensions of the shell elements surpass the necking length. This scaling ensures that the necking phenomenon can be evenly distributed along the length of the element, thus enabling a precise determination of structural failure. Due to the necking, the failure strain of shell elements is significantly affected by the size of the element. Consequently, for shell elements with disparate grid sizes, it is imperative to adjust the failure strain accordingly to maintain the fidelity of the simulation.

Adjustments to failure criteria are commonly grounded on the results of uniaxial tension tests. For instance, the critical equivalent plastic strain criterion as well as the RTCL failure criterion can be scaled using the relation presented in Equation (9):

$$\bar{\epsilon}_f = n + (\epsilon_n - n) \frac{t}{l} \quad (9)$$

Herein, ϵ_n represents the input value for fracture strain at the occurrence of $t/l = 1$, while n is the strain hardening exponent, indicative of the strain at the onset of necking during uniaxial tension. The grid size sensitivity of the failure strain is attributed to the necking phenomenon; given that the degree of necking varies under different stress states, it can be surmised that any adjustments for grid size sensitivity of the failure strain should also take into account the influence of the stress state (stress triaxiality).

This article adopts a correction method that considers both grid size sensitivity and stress state. The crux lies in altering the fracture strain ϵ_n , in Equation (9), and the diffuse necking condition n to become functions of the stress triaxiality η . Utilizing $\bar{\epsilon}_{f,cal}(\eta)$ to substitute ϵ_n in Equation (9), the following formula can be obtained:

$$\epsilon^*(\eta) = \bar{\epsilon}_N(\eta) + \left(\bar{\epsilon}_{f,cal}(\eta) - \bar{\epsilon}_N(\eta) \right) \frac{L_{cal}}{t_{cal}} \quad (10)$$

In this context, L_{cal} and t_{cal} represent the characteristic element size and the plate thickness, respectively, used when calibrating the failure criterion parameters. For the MMC failure criterion discussed in this paper, a solid element size of 0.1 mm is adopted for calibration, and the plate specimen thickness used for parameter calibration is 2 mm. Thus, it follows that $L_{cal} = 0.1$ mm and $t_{cal} = 2$ mm. By incorporating the stress-triaxiality-dependent failure fracture trajectory, denoted by $\bar{\epsilon}_{f,cal}(\eta)$, and the Swift local necking curve, denoted by $\bar{\epsilon}_N(\eta)$, into Equation (10), one can obtain the failure trajectory $\epsilon^*(\eta)$ for $t/L = 1$ shell elements, as depicted in Figure 4.

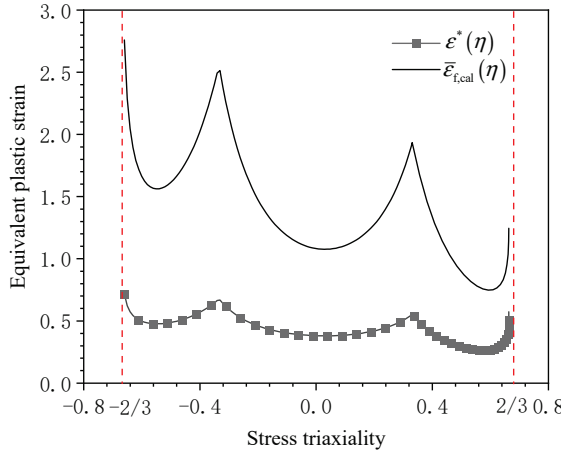


Figure 4. Calculate $\epsilon^*(\eta)$ according to $\bar{\epsilon}_{f,cal}(\eta)$.

Subsequently, through the inversion of Equation (10), the failure strain function that corresponds to any characteristic element length, denoted as $\bar{\epsilon}_f(L, t, \eta)$, can be derived, as demonstrated in Equation (11). With Equation (11), a series of the MMC failure criterion curves associated with different t/L values can be obtained through interpolation, as illustrated in Figure 5.

$$\bar{\epsilon}_f(L, t, \eta) = \bar{\epsilon}_N(\eta) + (\epsilon^*(\eta) - \bar{\epsilon}_N(\eta)) \frac{t}{L} \quad (11)$$

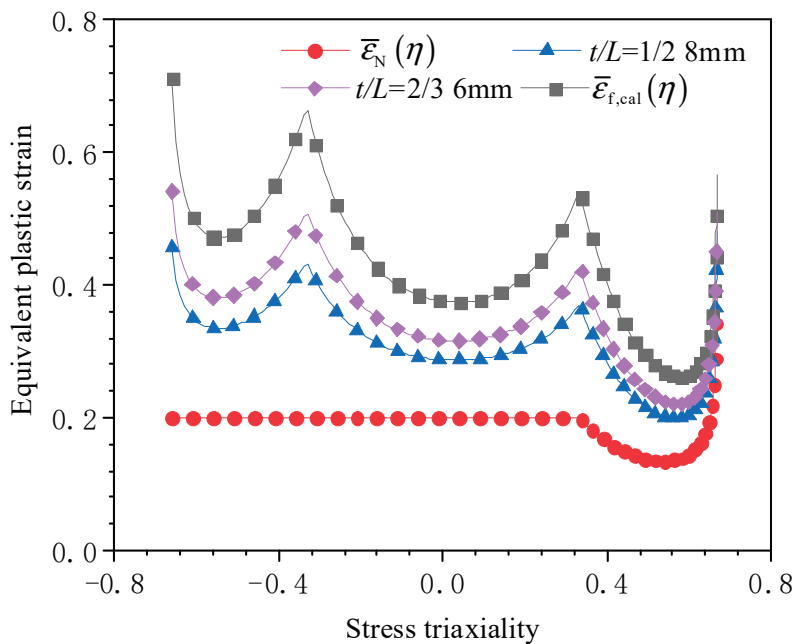


Figure 5. The MMC failure loci with different mesh sizes in $(\eta, \bar{\epsilon}_f)$ dimension.

It should be mentioned that the Swift necking curve typically exists in the $(\varepsilon_1, \varepsilon_2)$ space. The expression of the load stress ratio, represented by the symbol $\alpha = \sigma_2/\sigma_1$, can be described as follows, assuming that it stays constant:

$$\begin{cases} \varepsilon_1 = \frac{2(2-\alpha)(1-\alpha+\alpha^2)}{4-3\alpha-3\alpha^2+4\alpha} n \\ \varepsilon_2 = \frac{2(2\alpha-1)(1-\alpha+\alpha^2)}{4-3\alpha-3\alpha^2+4\alpha} n \end{cases} \quad (12)$$

In order to convert Equation (12), which exists within the $(\varepsilon_1, \varepsilon_2)$ space, to the curve $\bar{\varepsilon}_N(\eta)$ within the $(\eta, \bar{\varepsilon}_f)$ space as reflected in Equation (10) to Equation (11), the following transformations, represented by Equations (13)–(15), need to be applied to Equation (12).

Under the condition of plane stress, the equivalent plastic strain can be expressed as follows:

$$\bar{\varepsilon} = \frac{2}{\sqrt{3}} \sqrt{1 + \beta + \beta^2} \cdot \varepsilon_1 \quad (13)$$

In the equation, β represents the strain ratio, defined by $\beta = \varepsilon_2/\varepsilon_1$. Based on the von Mises criterion, there is a one-to-one correspondence between stress and strain under plane stress conditions. Consequently, the strain ratio β is related to the stress ratio α as follows:

$$\alpha = \frac{2\beta + 1}{\beta + 2} \quad (14)$$

Therefore, the stress triaxiality can also be obtained by calculating through the strain ratio.

$$\eta = \frac{1}{\sqrt{3}} \frac{1 + \beta}{\sqrt{1 + \beta + \beta^2}} = \frac{1 + \alpha}{\sqrt{1 - \alpha + \alpha^2}} \quad (15)$$

Based on Equations (13)–(15), the Swift instability curve in the $(\varepsilon_1, \varepsilon_2)$ space can be transformed into a curve $\bar{\varepsilon}_N(\eta)$ in the $(\eta, \bar{\varepsilon}_f)$ space. Since the Swift curve corresponds to a stress range between uniaxial tension and equi-biaxial tension ($1/3 \leq \eta \leq 2/3$), to facilitate scaling the MMC failure criterion throughout the entire range of plane stress states, $\bar{\varepsilon}_N(\eta) = n$ is selected within the $(-2/3 \leq \eta \leq 1/3)$ range, resulting in Equation (16). This is plotted in the $(\eta, \bar{\varepsilon}_f)$ space as shown in Figure 5.

$$\bar{\varepsilon}_N(\eta) = \begin{cases} n & -2/3 \leq \eta < 1/3 \\ \text{Swift necking curve} & 1/3 \leq \eta \leq 2/3 \end{cases} \quad (16)$$

2.3. The MMC Failure Criterion Embedding and Verification

2.3.1. Development of a VUMAT Subroutine

Abaqus, as a well-established general-purpose finite element software, offers 42 user subroutine interfaces that allow users to customize material properties, boundary conditions, special elements, and complex loads, as well as interfaces with other programs. The material subroutines UMAT and VUMAT enable users to implement their own algorithms and constitutive models of materials. In the domain of ship collision, the focus is often on the deformation process and residual deformation of structures under external loads. Thus, the implementation of the MMC failure criterion within finite element analysis is achieved using the VUMAT subroutine [21].

2.3.2. Simulation of Standard Test Specimens

To confirm the updated MMC failure criterion, quasi-static uniaxial tensile tests are designed and executed utilizing marine-grade high-strength steel, per the rules found in Metallic materials—Tensile testing at high strain rates—Part 1: Elastic-bar-type systems (GB/T 30069.1-2013). The tensile specimens and their dimensions are depicted in Figure 6. To ensure the reliability of the test results, three identical specimens are fabricated and three sets of material tensile tests are performed. To guarantee that the specimen is loaded under

quasi-static conditions, the test equipment's tensile speed is controlled at 1.0 mm/min. The load experienced by the specimen and the elongation during the tensile process are recorded by the sensors integrated within the testing apparatus, enabling the acquisition of the load–displacement curve.

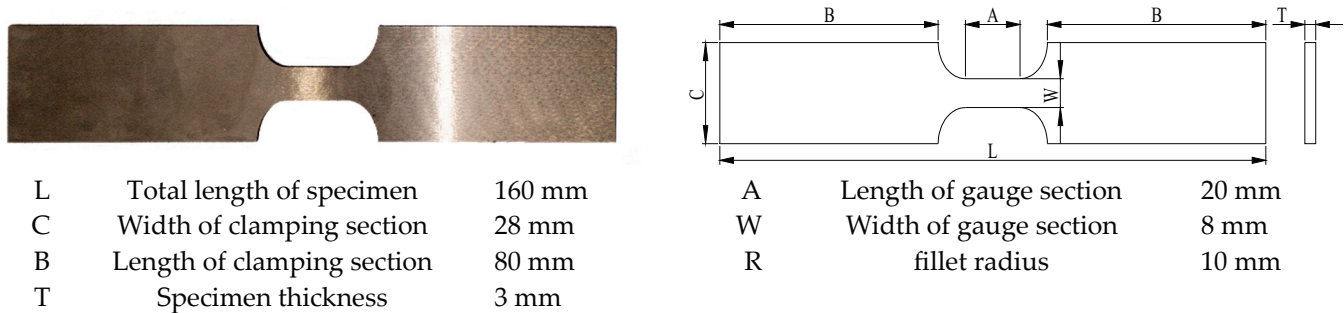


Figure 6. Quasi-static tensile test specimen size diagram.

The load–displacement curves obtained from the experimental results were converted into engineering stress–strain curves based on the effective cross-sectional dimensions of the actual specimens, as shown in Figure 7a. The conversion of the engineering stress–strain curves to true stress–strain was achieved by employing the composite relation curve of weighted averages proposed by Ling [22], as depicted in Figure 7b. The resulting parameters for the marine-grade high-strength steel are presented in Table 3.

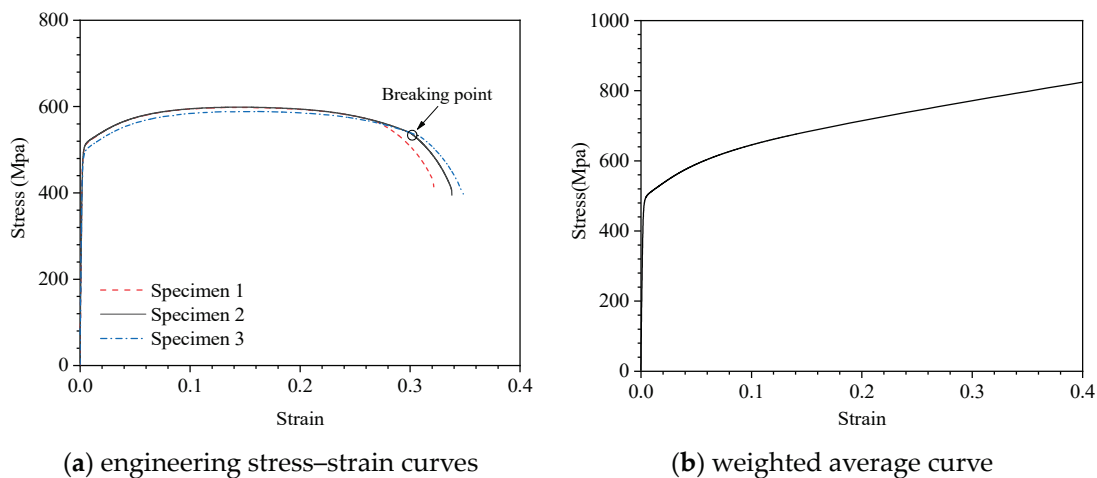


Figure 7. Stress–strain curve for marine-grade high-strength steel.

Table 3. Marine-grade high-strength steel material parameters.

Density $\rho \text{ kg/m}^3$	Elastic Modulus E Gpa	Poisson Ratio	Yield Stress Mpa	Failure Strain
7850	210	0.3	470	0.28

For finite element simulation based on material testing, a finite element model is established for the middle parallel section of the specimen. Due to size limitations of the specimen, this part is discretized into finite element models of merely two mesh sizes, controlling the central element dimensions straining in the tensile direction to be 3 mm and 4 mm, respectively. The finite element model is illustrated in Figure 8. Reference points are established at the midpoints of both ends of the parallel section and are coupled to the respective ends; one reference point is rigidly constrained while the other point is allowed

to release displacement in the tensile direction only. The displacement loading rate and mass scaling are maintained as consistent with the preceding section.

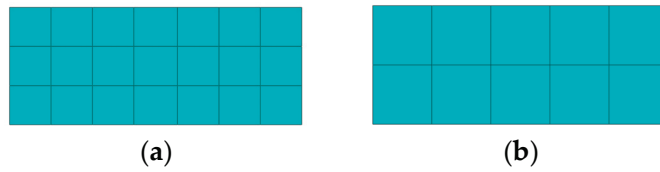


Figure 8. Finite element model of parallel section of standard specimen: (a) 3 mm × 2.7 mm; (b) 4 mm × 4 mm.

Due to the utilization of standard specimens, the engineering stress–strain curve can be deduced from the load–displacement graph. In Figure 9, a comparison of the engineering stress–strain curves for two different mesh sizes with the experimental data reveals a good agreement, which suggests that the MMC failure criterion can accurately predict the behavior of the standard tensile specimens addressed in this study.

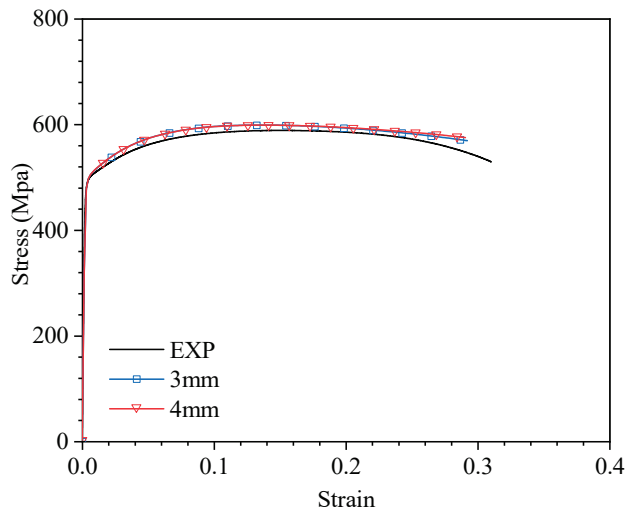


Figure 9. Comparison between experimental data and numerical force versus displacement curves.

3. Application of the MMC Failure Criterion in the Simulation of Ship Plate Frame Structure Collisions

3.1. Experimental Setup

The collision impact tests were conducted on the 50 kJ drop hammer impact testing apparatus at Jiangsu University of Science and Technology. This apparatus consists of counterweights, guide rails, an accelerometer, a hammer head, a buffer, displacement sensors, and supports. The main frame of the test apparatus is 6.3 m high, with a wedge-shaped hammer head width of 420 mm. The hammer head can be lifted to a maximum height of 3.7 m, achieving a maximum impact speed of 8.5 m/s. The maximum mass of the hammer head and counterweight system is 1350 kg, capable of generating a maximum impact energy of approximately 50 kJ. An accelerometer is affixed to the hammer body to calculate collision forces generated during the drop hammer test, while displacement sensors located on both sides of the support record the positional changes of the hammer body. The test apparatus and the wedge-shaped hammer head are depicted in Figure 10.

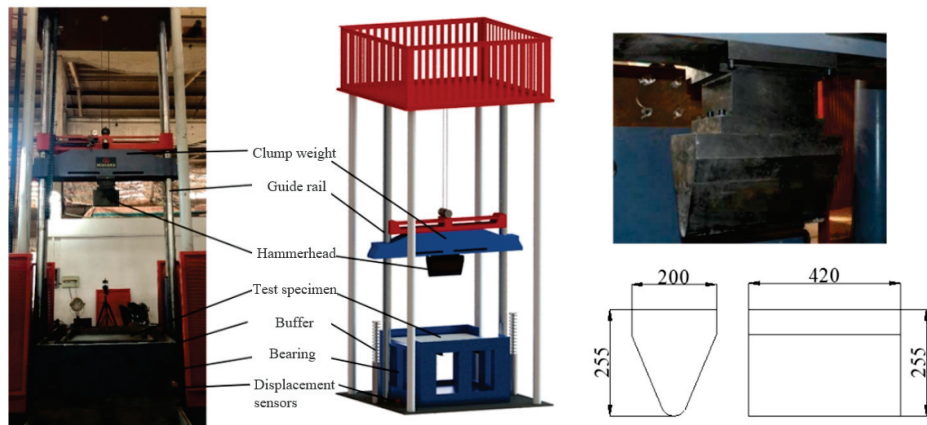


Figure 10. Fifty-kilojoule drop-weight impact test apparatus with a wedge-shaped hammer diagram (unit: mm).

3.2. Test Specimen Design

Based on the collision scenario depicted in Figure 11, which involves a little contact between the bow and the plate frame, we constructed the experiment. The plate frame specimens used in the experiments were fabricated from high-strength marine steel, with dimensions as shown in Table 4. A flange, 30 mm in width, was retained around the periphery of the specimen's faceplate. The plate frame is inserted into the tooling through the panel's reserved flange, and the tooling is welded to the plate frame, dimensionally corresponding to the actual object as depicted in Figure 12. The tooling is sufficiently rigid to be considered a rigid body and is attached to the foundation. Furthermore, it is important to minimize the heat effect of the plastic deformation of the structure during the welding process. Additionally, the tooling position needs to be adjusted to guarantee that the hammer head and the plate frame are centered. The high carbon chromium bearing steel (GCr15) impact hammer head is wedge-shaped and attached to the hammer body with a counterweight block to generate impact energy.

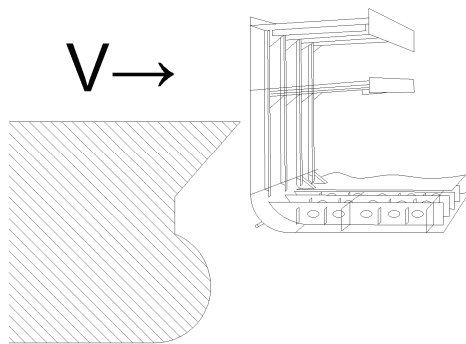


Figure 11. Diagrammatic representation of the crash site.

Table 4. The size of specimens (unit: mm).

Specimen Name	Quilting Material Spacing	Falling Weight Height	Slab Dimension	Bone Size	Bone Spacing	Truss Size
Plate frame structure	1000	2000	1560 × 1560 × 4	6# bulb flat	390	$\perp \frac{4 \times 160}{5 \times 80}$

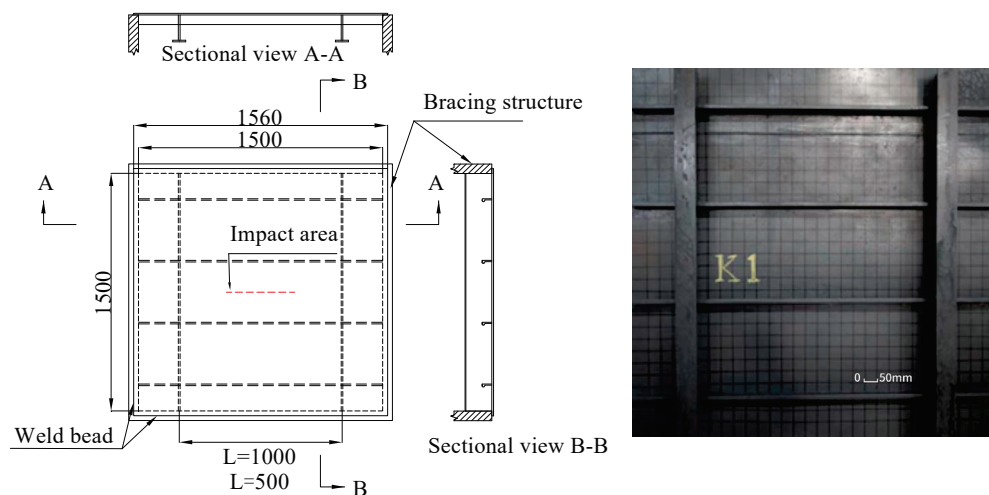


Figure 12. The dimension and picture of plate frame structure (Unit: mm).

3.3. Finite Element Model

Finite element models of the plate frame structure were constructed for drop hammer impact, as shown in Figure 13a. Models comprised both the plate frame structure and the impactor. Following the hammer head dimensions provided in Figure 14, a rigid hammer head was created in the finite element simulation with a mass of 1350 kg. The mesh size for the contact area between the hammer head and the panel was set to 4 mm, and the plate frame structures model is depicted in Figure 13b. In the finite element software, a reference point was added to the rigid hammer head, along with a mass of 1350 kg, and only the degrees of freedom in the U_z direction were released.

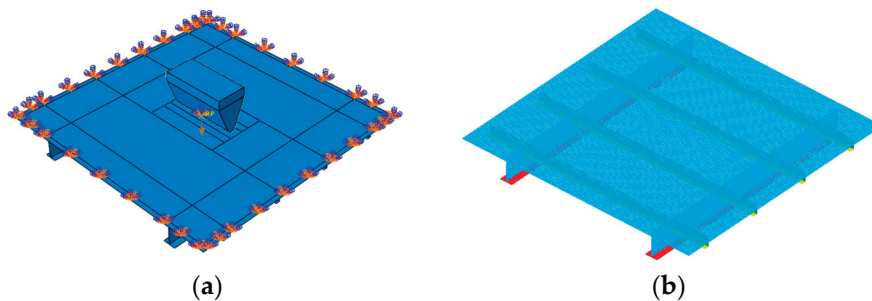


Figure 13. The finite element model of drop indenter impact scene. (a) Geometric model; (b) finite element model.

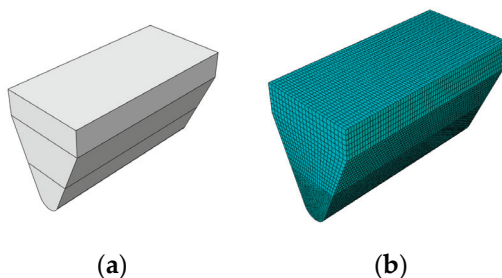


Figure 14. The geometry and finite element model of wedge indenter. (a) Geometric model; (b) finite element model.

In simulation calculations, the boundary conditions of the plate frame model should align as closely as possible with the actual experimental setup. The outer boundaries of the plate frame are welded to the support structure and can be considered as rigidly fixed. Therefore, the simulation involves applying fixed constraints to the external edges of the

plate frame ($U_x = U_y = U_z = UR_x = UR_y = UR_z = 0$) [23]. The inner boundaries of the plate frame are supported by the support structure; thus, the simulation restricts their displacement in the vertical direction ($U_z = 0$).

The material model for the metal plate frame is the same as described in Section 2.3.2, which is marine-grade high-strength steel. To account for the material's strain rate effects, the Cowper–Symonds (C-S) model is employed to describe its strain rate effect. This model is characterized by its applicability within the medium to low strain rate range and is widely used in ship collision simulation calculations. The expression for the Cowper–Symonds model is as follows:

$$\frac{\sigma_d}{\sigma_s} = 1 + \left(\frac{\dot{\epsilon}}{D} \right)^{q-1} \quad (17)$$

The dynamic yield stress is denoted by σ_d , while σ_s represents the static yield stress, which can be obtained through uniaxial tensile tests. $\dot{\epsilon}$ denotes the equivalent plastic strain rate, and D and q are the Cowper–Symonds (C-S) constants, obtainable through fitting with dynamic testing data. For marine-grade high-strength steel, high-speed tensile tests at various strain rates were performed, resulting in material engineering stress–strain curves at different strain rates as shown in Figure 15. It can be observed that, with increasing strain rates, the flow stress of the marine structural steel continuously increases; furthermore, the toughness of this type of marine structural steel also increases with strain rate, a characteristic that significantly differs from low-carbon steels. The yield stresses from high-speed tensile tests at different strain rates were extracted as presented in Table 5. Fitting these data with the C-S constitutive model, values of $D = 678,000$ and $q = 5.261$ are obtained, as illustrated in Figure 16. The parameters for the marine-grade high-strength steel material are presented in Table 6.

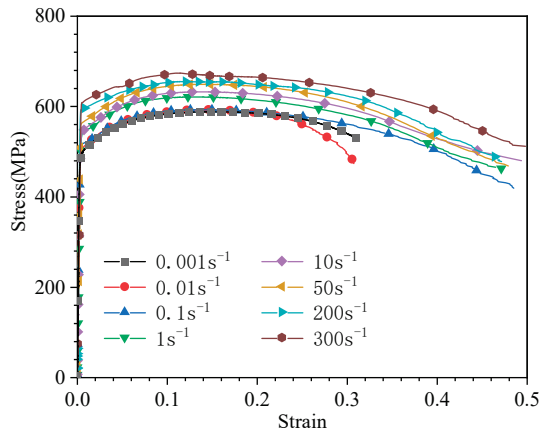


Figure 15. Strain–stress curves at different strain rates.

Table 5. Dynamic and static mechanic behavior of marine high-strength steel.

strain rate/s ^{−1}	0.001	0.01	0.1	1	10	50	200	300
yield stress/MPa	489	501	516	540	541	561	592	608

Table 6. Marine-grade high-strength steel material parameters.

Density ρ kg/m ³	Elastic Modulus E Gpa	Poisson Ratio	Yield Stress Mpa	Failure Strain	D	q
7850	210	0.3	470	0.28	678,000	5.261

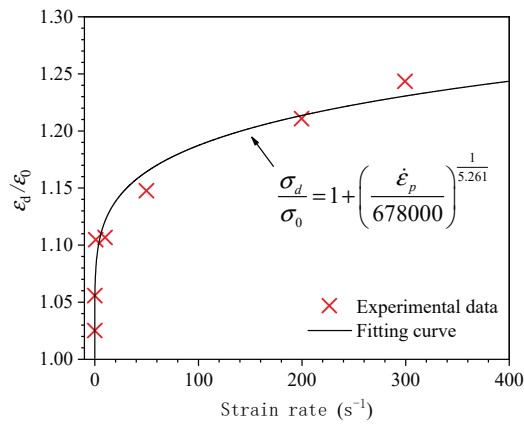


Figure 16. Cowper–Symonds model fitted by dynamic tensile data.

The contact type between the plate frame and hammer head is designated as “general contact”, with the contact properties’ normal behavior set as “hard contact” and tangential behavior chosen as “penalty”, with a friction coefficient of 0.3 as per reference [24].

3.4. Comparison of Simulation and Test Results

3.4.1. Damage Deformation

Figure 17 presents the results of the plate frame experiments under drop-weight impact loading compared to the damage contour maps computed using the MMC failure criterion. It is observable that panel indentation occurred around the impact area; due to the support provided by the stiffeners, the indented area was confined within the boundaries framed by the stringers and the web members, resulting in an elliptical indentation whose major axis aligns with the direction of the wedge-shaped head’s length. The deformation of the panels on either side of the impact mid-region was less significant, a phenomenon attributed to the initial shear failure at the panel edges beside the wedge-shaped impactor, which inhibited the efficient transmission of subsequent impact energies outward. The structure and load are symmetrical with respect to the longitudinal axis, but the response is not symmetrical, and this phenomenon occurs because the shape of the spherical flat steel in the plate frame is asymmetrical, so the resulting response is also asymmetrical.

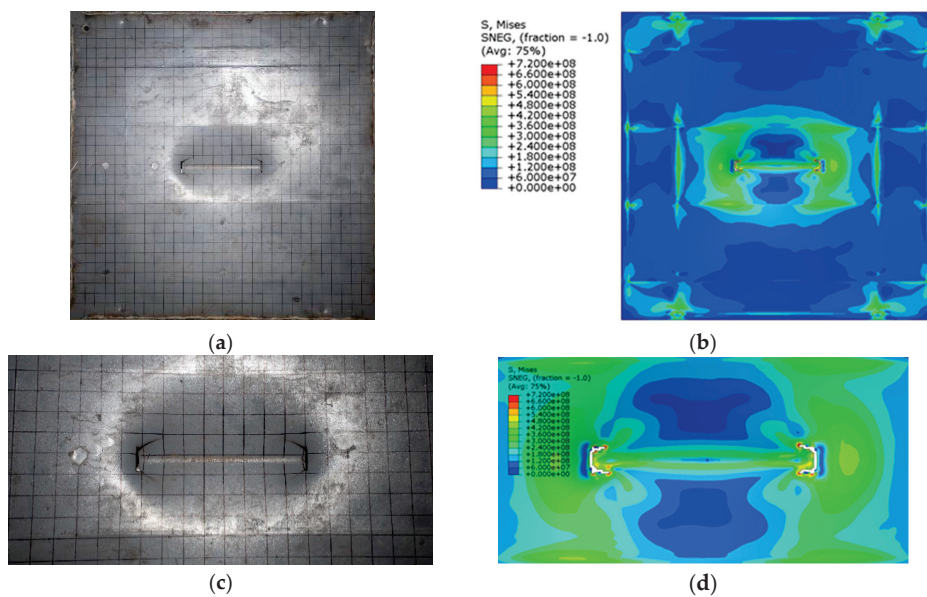


Figure 17. Contour of von Mises stress of plate frame under drop hammer collision condition. (a) Overall damage image; (b) overall contour of von Mises stress; (c) localized damage image; (d) localized contour of von Mises stress.

From the perspective of the damage state, the drop-weight impact loading tests revealed a “crescent-shaped” rupture in the plate frame structure, which showed good correspondence with the simulation results. Further analysis of the collision history contour maps, as indicated in Figure 18, reveals that the initial impact led to straight shear fracture, and the geometric continuity disruption at the fracture caused the load-bearing state of the structure at the rupture to evolve from a vertical shear to a coupled tearing of shear and in-plane tension. As a result, the fracture direction shifted from a parallel cut relative to the edge of the wedge head to a more obliquely oriented incision toward the direction of the wedge head’s length. With progressive deepening of the fracture, shear effects diminished while tensile effects increased, indicating a gradual inclination of the rupture direction toward the wedge head’s longitudinal direction.

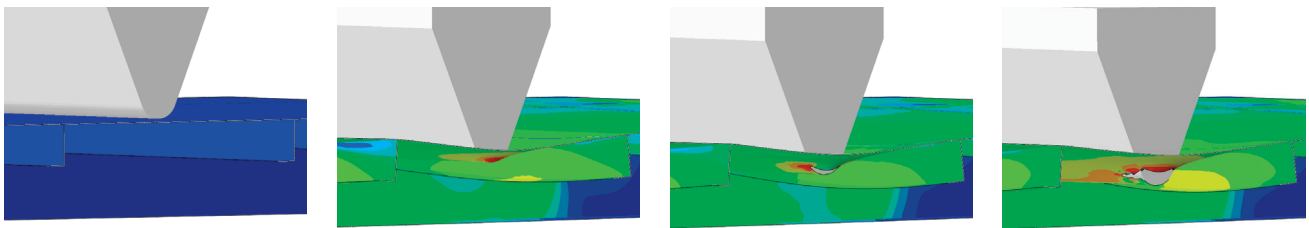


Figure 18. Damage deformation process of plate frame.

The measurement of plastic strain is conducted using the measure points indicated in Figure 19, and the overall deformation trend of the experiment and the three calculation methods is compared. It is evident that the trend of the MMC failure criterion is more similar to the experiment than the other two, and that the overall deformation trend of the three calculation methods is similar to the experimental results.

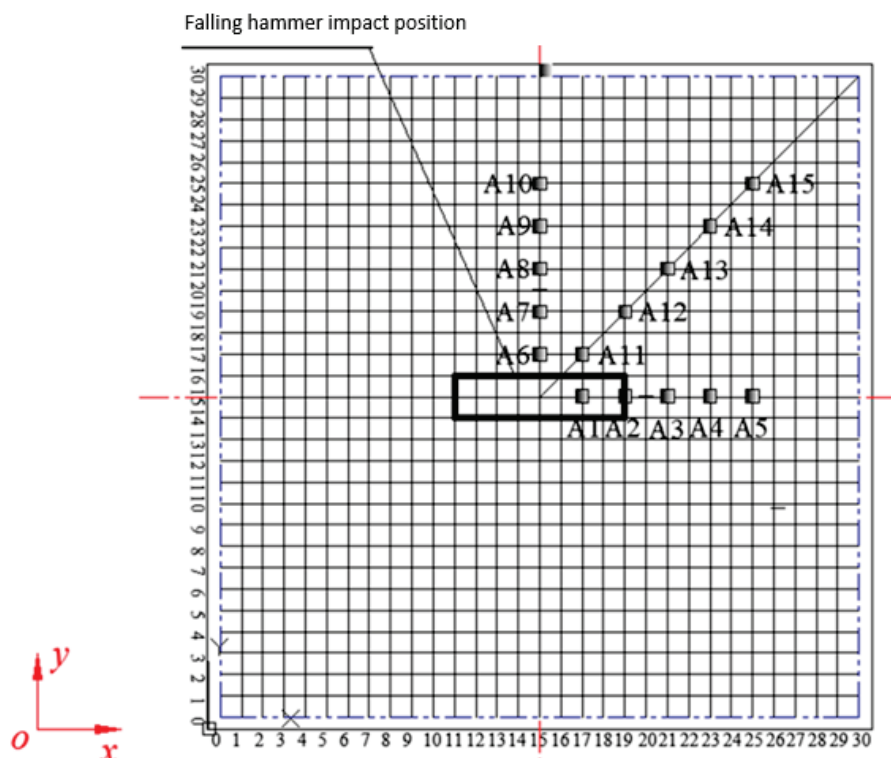


Figure 19. The positions of measuring points (unit: mm).

3.4.2. Force–Time Curve of Collision

The impact force–time curve obtained from the simulation using the MMC failure criterion is illustrated in Figure 20. It is evident that the simulated impact force–time curve aligns well with the experimental results, demonstrating a consistent overall trend. The simulation accurately predicts the early stage of the experimental curve’s impact force–time profile; however, some discrepancies are present during the middle and later stages. Concerning the peak impact force encountered during the collision, experimental measurements record a peak value of 0.547 mega-newtons (MN). In contrast, simulation-derived data present a slightly higher peak at 0.5854 MN. This indicates a deviation of 7.02% from the experimentally determined figure. Notwithstanding this difference, the margin of error falls within an acceptable range for such simulations in marine collision analysis [23,25], substantiating the simulation’s overall validity and precision. However, it was observed that the simulated impact duration exceeded the experimental impact time, which may be attributed to the fact that the boundary conditions of the specimen in the experiment are not ideally rigidly fixed.

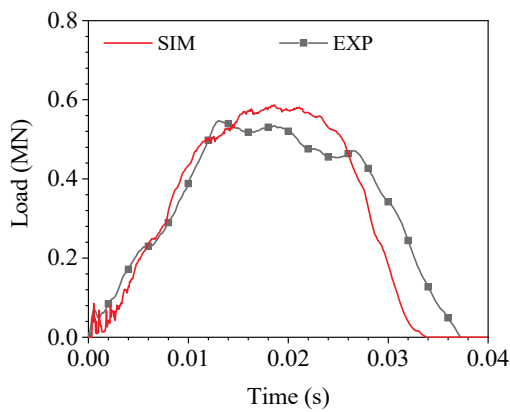


Figure 20. The force–time curve calculated by revised MMC failure criterion.

4. Discussion

The foregoing discussion detailed the calibration of the MMC metal failure criterion, which demonstrated favorable outcomes when applied to material experiments as well as drop-weight impact test simulations. Building upon this, simulation studies were conducted on plate frame structure collision scenarios under various failure criteria and mesh sizes to further ascertain the robustness of the MMC failure criterion.

4.1. Common Fracture Failure Criteria in Ship Collision Simulation

Over the past few decades, with the widespread application of the finite element method (FEM) in the field of ship collisions, the failure criteria used in ship collision simulations have become a hot topic of interest among researchers internationally and have seen significant developments. To further verify the accuracy of the MMC failure criterion, two commonly applied criteria in ship collision simulations were selected for comparative analysis: the equivalent plastic strain (EPS) criterion and the RTCL failure criterion.

4.1.1. Equivalent Plastic Strain Criterion

The critical equivalent plastic strain criterion, also known as the constant plastic strain criterion, defines that material undergoes localized fracturing when the value of the equivalent plastic strain reaches a critical level, as shown in Equation (18):

$$\bar{\varepsilon} = \frac{\sqrt{2}}{3} \sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2} = \bar{\varepsilon}_f \quad (18)$$

In the formula, $\bar{\epsilon}$ represents the equivalent plastic strain, while $\epsilon_1, \epsilon_2, \epsilon_3$ signify the first, second, and third principal strains, respectively. $\bar{\epsilon}_f$ denotes the critical equivalent plastic strain. This critical value is often referred to as the maximum plastic strain or critical strain, and it is typically determined through uniaxial tensile tests. However, this failure criterion overlooks the influence of stress states on material failure. For instance, it assumes that materials can fracture under pure compression, which contradicts empirical experimental phenomena. Since the failure strain of this criterion is derived from uniaxial tensile testing, it is sensitive to mesh size. Therefore, when simulating with this failure criterion, it is important to scale the failure strain according to the actual mesh size used.

4.1.2. RTCL Failure Criterion

Rice and Tracey [26], building upon the foundation of micromechanics of damage, have derived an expression for the Rice–Tracey ductile failure criterion by studying the growth behavior of internal defects in materials (such as spherical voids) under loading. The expression for the Rice–Tracey criterion is presented in Equation (19). This criterion is capable of accurately predicting the growth of voids and is applicable over a wide range of triaxialities.

$$D = \int_0^{\bar{\epsilon}_f} \exp(1.5\eta) d\bar{\epsilon} \quad (19)$$

Subsequently, Cockcroft and Latham [27] proposed a failure criterion based on the equivalent plastic strain corrected for the maximum principal stress, as shown in Equation (20). The Cockcroft–Latham criterion exhibits good predictive capabilities for ductile shear failure within a low range of stress triaxiality.

$$D = \int_0^{\bar{\epsilon}_f} \sigma_1 d\bar{\epsilon} \quad (20)$$

Tornqvist [5], by integrating the Rice–Tracey and Cockcroft–Latham criteria along with a cutoff value where fracture failure does not occur, introduced the RTCL criterion suitable for a broader range of stress states, as presented in Equations (21) and (22):

$$D = \frac{1}{\bar{\epsilon}_f} \int_0^{\bar{\epsilon}_f} \begin{cases} 0 & \eta < -\frac{1}{3} \\ \frac{\sigma_1}{\bar{\sigma}} d\bar{\epsilon}^p & -\frac{1}{3} \leq \eta \leq \frac{1}{3} \\ \exp\left(\frac{3\eta-1}{2}\right) d\bar{\epsilon}^p & \eta > \frac{1}{3} \end{cases} \quad (21)$$

$$\frac{\sigma_1}{\bar{\sigma}} = \frac{2 + 2\eta\sqrt{12 - 27\eta^2}}{3\eta + \sqrt{12 - 27\eta^2}} \quad (22)$$

In the formula, D represents the damage factor, which leads to the elimination of the element when $D = 1$ reaches a critical value. $\bar{\epsilon}^p$ denotes the equivalent plastic strain, η is the stress triaxiality, and $\bar{\epsilon}_f$ signifies the fracture strain that also serves as the sole input parameter for the RTCL failure criterion. This parameter can be determined through uniaxial tensile testing. The RTCL criterion is relatively sensitive to mesh size. Therefore, it is essential to scale the failure strain obtained under uniaxial tensile conditions to reduce its mesh size sensitivity. The fracture scaling law is expressed as shown in Equation (23):

$$\epsilon_f = \epsilon_r + (\epsilon_n - \epsilon_r) \frac{t_e}{l_e} \quad (23)$$

Within this context, ϵ_r corresponds to the strain at which material diffuse necking occurs, and ϵ_n is the fracture strain under uniaxial tensile conditions when $t_e/l_e = 1$. Here, t_e represents the element thickness, and l_e denotes the element size. By combining Equations (22) and (23), one can derive the RTCL failure criterion, which is related to mesh size and stress triaxiality.

4.2. Comparative Analysis of Impact Responses of Stiffened Panels under Drop-Weight Collisions Using Different Failure Criteria

The stress damage contour plots obtained using the two failure criteria were compared with the damage photos from the plate frame test, as shown in Figures 21 and 22. It is observable that the stress levels and distributions calculated by the three failure criteria are relatively consistent. Among these, the failure criteria that take into account the state of stress, such as RTCL and MMC, are capable of accurately predicting the “crescent-shaped” rupture in the plate frame panels. In contrast, the EPS failure criterion, due to its lack of consideration for the relationship between the failure strain of elements and the state of stress, can only predict the shear rupture at the initial contact between the wedge-shaped hammer head and the panel. It is completely unable to foresee the damage propagation at both ends of the initial rupture.

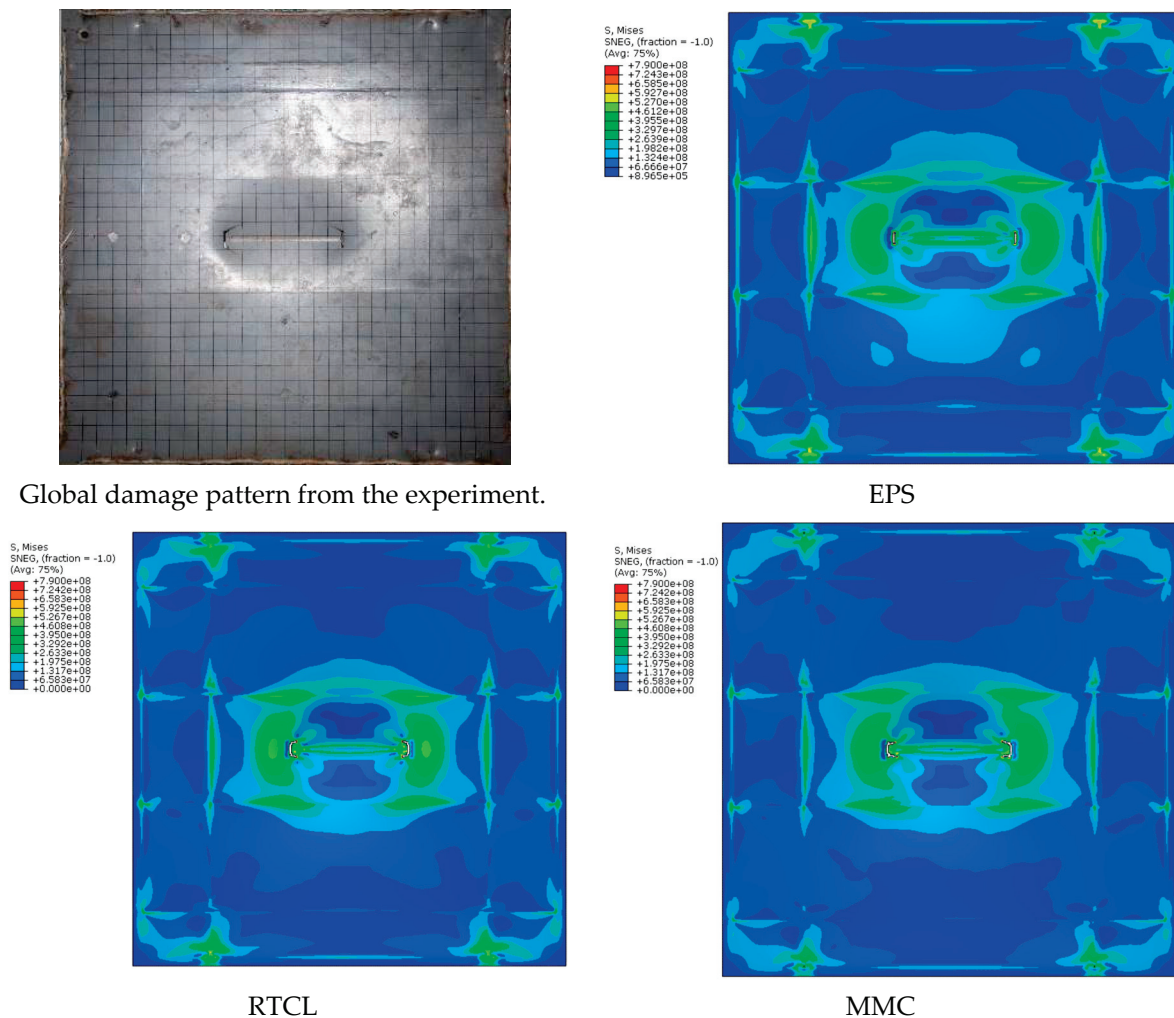


Figure 21. Damage prediction of different failure criteria for drop hammer collision conditions (overall).

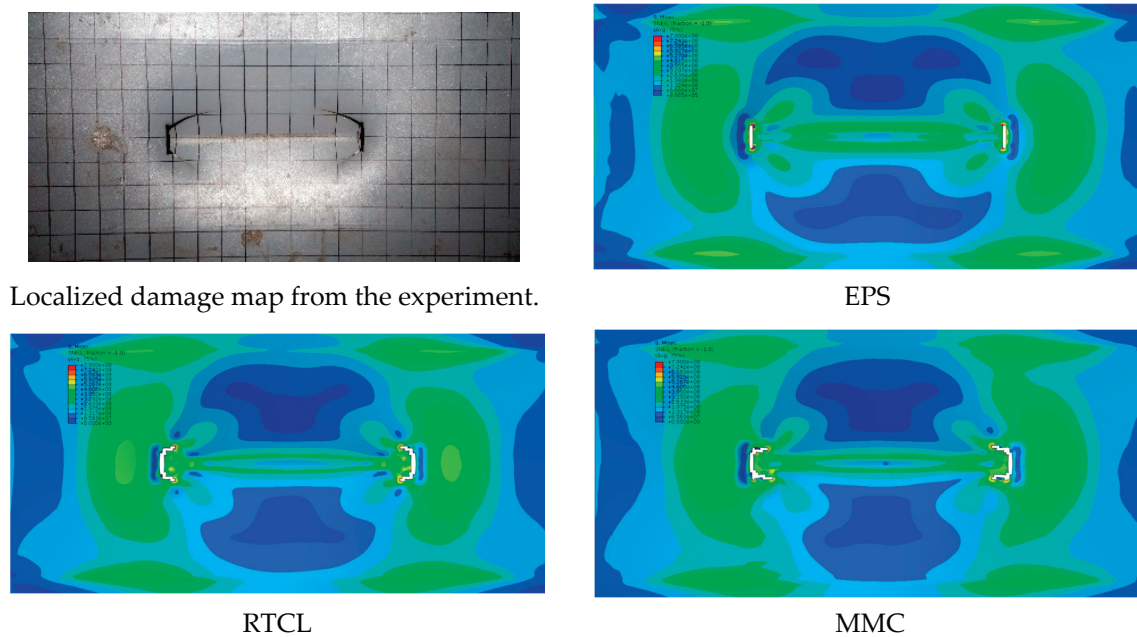


Figure 22. Damage prediction of different failure criteria for drop hammer collision conditions (local).

The simulation results for the EPS, RTCL, and MMC failure criterion were compared with experimental data, as shown in Figure 23. Since all other settings are consistent across the simulations except for the failure criteria, it is observed that the three simulation curves for each data set coincide completely during the initial phase of the collision. It is only when element failure occurs that the EPS and RTCL curves begin to diverge from the MMC curve. The peak collision force and the termination time of the collision force indicates that the MMC failure criterion provides a closer prediction of the collision force–time curve compared to the experimental results. This demonstrates the effectiveness of the MMC failure criterion.

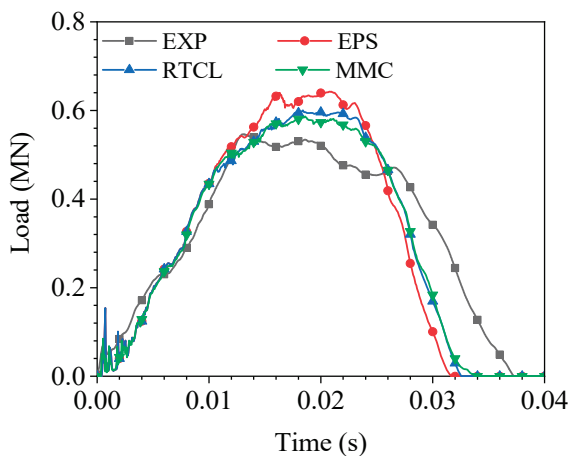


Figure 23. The force–time curve calculated by different failure criteria.

A comparative analysis of collision force peaks obtained from simulations using EPS, RTCL, and the modified MMC failure criterion against experimental peak values is presented in Table 7. It can be observed that the peak collision force error for the MMC failure criterion is 7.02%; for the RTCL failure criterion, it is 9.76%; and, for the EPS failure criterion, it is 17.48%. A comparative analysis of collision energy obtained from simulations using EPS, RTCL, and the modified MMC failure criterion against experimental collision energy values is presented in Table 8. It can be observed that the collision energy error for

the MMC failure criterion is 0.02%; for the RTCL failure criterion, it is 0.20%; and, for the EPS failure criterion, it is 2.11%. The simulation results utilizing the MMC failure criterion demonstrate smaller errors compared to those using the EPS and the RTCL failure criteria, further substantiating the effectiveness and superiority of the MMC failure criterion.

Table 7. Comparison of simulation collision force peak with experimental data.

Failure Criterion	Collision Force Peak (MN)	Error (%)
EXP	0.547	0
MMC	0.5854	7.02
RTCL	0.6004	9.76
EPS	0.6426	17.48

Table 8. Comparison of simulation collision energy with experimental data.

Failure Criterion	Collision Energy (KJ)	Error (%)
EXP	22.51	0
MMC	22.51	0.02
RTCL	22.46	0.20
EPS	22.03	2.11

4.3. Mesh Size Sensitivity Study of the MMC Failure Criterion

To validate the accuracy of the mesh sensitivity correction for the MMC failure criterion, a mesh size sensitivity analysis was conducted for the simulation of stiffened plate collisions. Given that the purpose of this work is to examine a small breach in the event of a minor ship accident and that a bigger mesh size is not suitable for accurately characterizing a smaller breach, the original geometric model was discretized into meshes of 4 mm, 6 mm, and 8 mm sizes, and finite element calculations were performed using the MMC failure criterion with corrected mesh sensitivity. The results of the mesh discretization are shown in Figure 24.

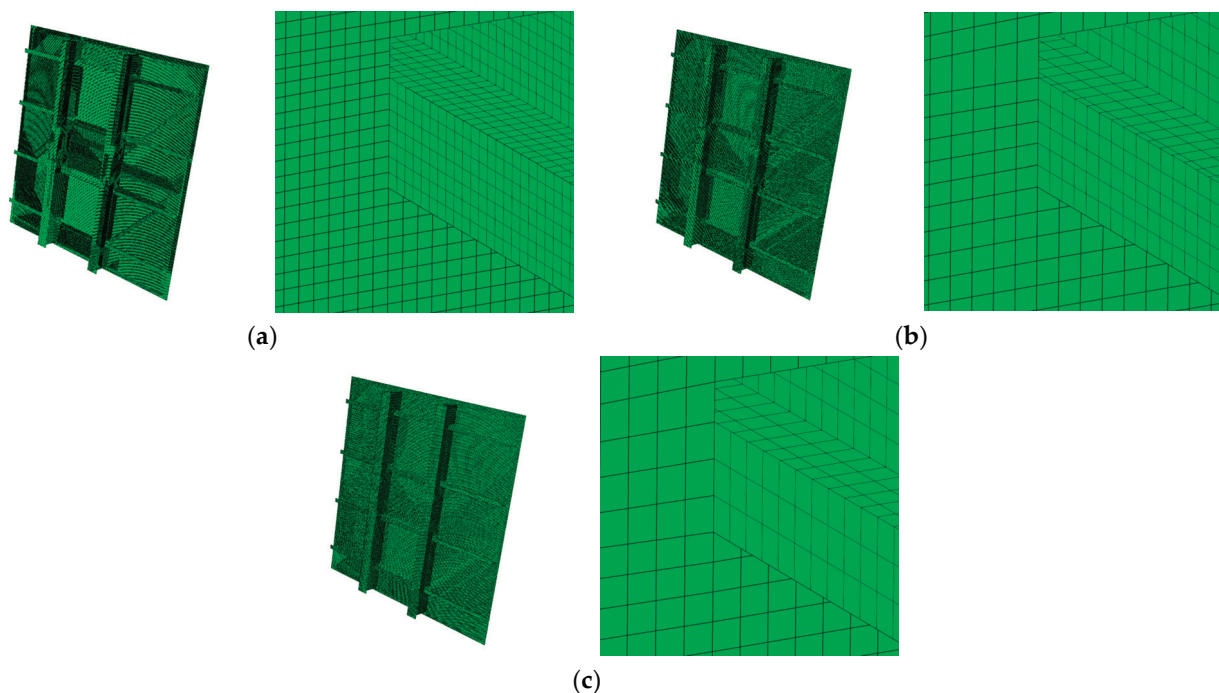


Figure 24. Finite element models of stiffened panels with different mesh sizes. (a) Mesh feature length of 4 mm; (b) mesh feature length of 6 mm; (c) mesh feature length of 8 mm.

The simulated results of the panel frame collision under different mesh sizes are depicted in Figures 25 and 26. It is observable that the overall deformation patterns of the panel frame are essentially consistent across the three mesh sizes, with the central impact area exhibiting a downward dent, and the large deformation zones of the panel being constrained by the stiffeners. The distribution of stress is largely uniform, with high-stress regions concentrated around the hammer impact tearing location. The morphology of the panel breaches is broadly similar, with each mesh size capable of replicating the initial shear breaches at both ends caused by the wedge-shaped hammer head. However, due to the influence of mesh size, although the 6 mm and 8 mm meshes can also simulate the subsequent tearing breaches, the experimentally observed breaches exhibit a “crescent” shape with elongated ends, which the coarser mesh sizes fail to accurately represent regarding the directional propagation of the tears. Therefore, employing a 4 mm mesh size can more accurately simulate the deformation and damage of the panel frame under the hammer impact conditions.

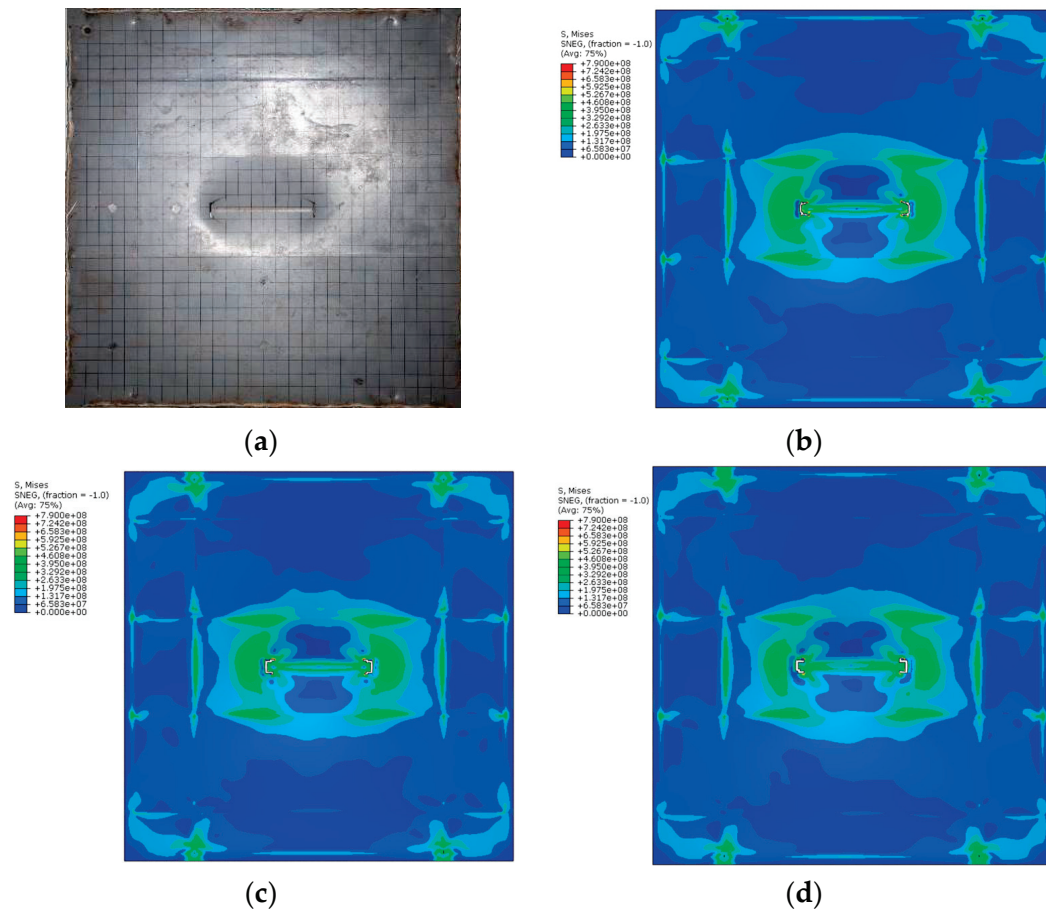


Figure 25. The overall contour of von Mises stress of different mesh size models under drop hammer collision conditions. (a) Test damage diagram; (b) 4 mm finite element model; (c) 6 mm finite element model; (d) 8 mm finite element model.

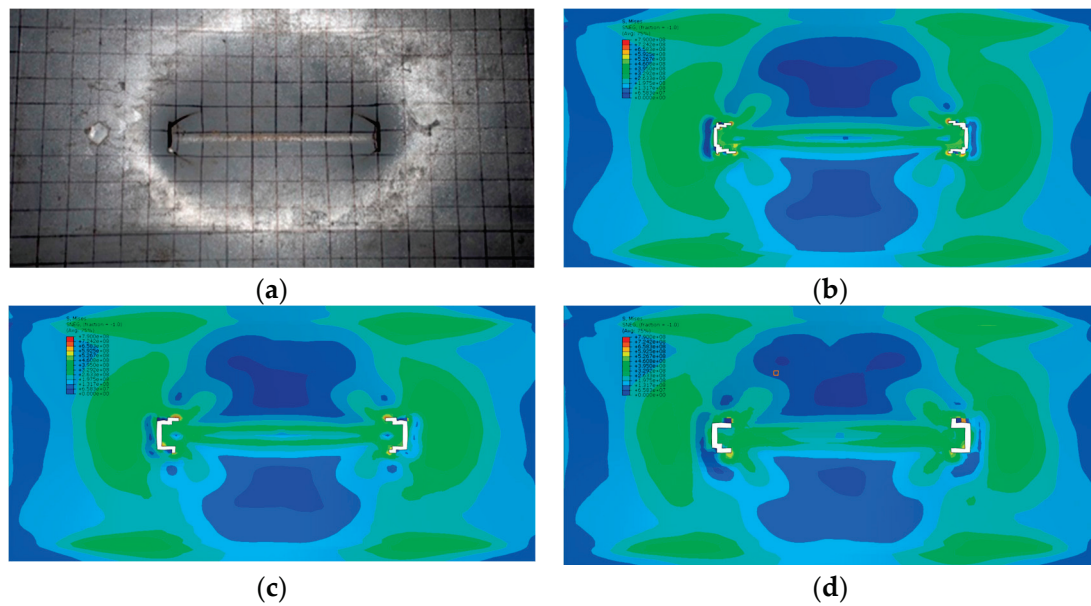


Figure 26. Local contour of von Mises stress of different mesh size models under drop hammer collision condition. (a) Test damage diagram; (b) 4 mm finite element model; (c) 6 mm finite element model; (d) 8 mm finite element model.

A comparison of the collision force curves under different mesh sizes with experimental results is shown in Figure 27. It can be observed that, in the early stage of the curves, before any damage to the panel frame occurs, the collision force–time curves for different mesh sizes are in good agreement. After the onset of damage, the three curves exhibit some differences, but, overall, they are closely aligned. It can be concluded that the finite element results for various mesh sizes are convergent. This outcome substantiates the effectiveness of the mesh size sensitivity correction applied to the MMC failure criterion.

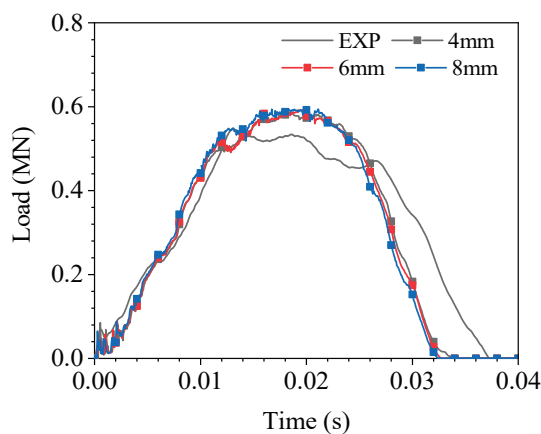


Figure 27. The force–time curve corresponding to different mesh models.

5. Conclusions

This study uses simulations and tests to calibrate and apply the MMC failure criterion to the crashworthiness of hull plate structures. The ship collision simulation's issue with the input of metal materials has been resolved, where a typical plate frame structure is selected for collision analysis based on the collision mode of minor collisions of ships, calibrating the MMC failure criterion for marine high-strength steel and furthering this by implementing a mesh sensitivity correction that considers the stress state. Through panel frame drop hammer impact experiments and tensile test simulations, the superiority of the modified MMC failure criterion was confirmed. This establishes a theoretical foundation

for the safety design and assessment of hull beam structures going forward. The main conclusions are as follows:

(1) Utilizing MATLAB optimization tools, the three-dimensional fracture surface and two-dimensional fracture trajectory of the MMC failure criterion were fitted, and a mesh size sensitivity correction considering the influence of stress states was conducted. By including the MMC failure criterion into the Abaqus VUMAT subroutine and contrasting the outcomes of the simulation and experiment, the validity of the modified MMC failure criterion was confirmed.

(2) Simulations using the MMC failure criterion were carried out for the drop hammer collision test on the stiffened plate of ship structures, where the damage response mode under impact from a wedge-shaped head was studied, and the deformation mechanism was analyzed. A comparison between experimental and simulated damage deformation states and collision force–time curves verified the reliability of the MMC failure criterion.

(3) Simulations of the panel frame structure of a ship subjected to drop hammer collision were conducted using different failure criteria. The damage deformation contour plots and collision force curves were compared, indicating that the stress-state-aware failure criterion could more accurately simulate the deformation pattern of the breaches. Taking into account both the breach and the collision force–time curves, the MMC failure criterion showed a closer agreement with experimental results compared to the other two criteria.

(4) A mesh size sensitivity analysis utilizing the modified MMC failure criterion was performed. While damage states differed under various mesh sizes, the collision force–time curves were largely consistent, and the simulation results converged. This confirms the reliability of the correction method described in this paper.

(5) Under the experimental and simulation conditions, the MMC failure criterion in this study agrees well with the experiment; however, it is unclear how changing other parameters, like plate thickness and hammer shape, will impact the performance of various fracture standards. Future research may take this area of study as its direction.

The MMC failure criterion simulation and test conducted in this study serve as a guide for ship structure safety. A typical unit structure in ships is the stiffened plate, whose damage can serve as a model for actual ship damage in complicated collision scenarios and offer recommendations for the crashworthiness of ship designs. This work does not provide a comprehensive picture of the effect of the reinforced plate because it only discusses one type of plate thickness, hammer head, impact speed, impact strength, and other instances. Further tests testing various combinations of plate thickness, hammer head, impact velocity, and impact strength are necessary to have a complete understanding of the behavior of reinforcing plates.

Author Contributions: Writing—original draft, S.L.; formal analysis, S.L.; conceptualization, H.L.; writing—review and editing, K.L.; visualization, S.Z. and C.L. software, H.L.; validation, Y.L. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Natural Science Foundation of China (Grant No. 52171311; No. 52271279).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Alsos, H.; Hopperstad, O.; Törnqvist, R.; Amdahl, J. Analytical and numerical analysis of sheet metal instability using a stress based criterion. *Int. J. Solids Struct.* **2008**, *45*, 2042–2055. [CrossRef]
2. Hill, R. On discontinuous plastic states, with special reference to localized necking in thin sheets. *J. Mech. Phys. Solids* **1952**, *1*, 19–30. [CrossRef]

3. Minorsky, V. *An Analysis of Ship Collisions with Reference to Protection of Nuclear Power Plants*; Sharp (George G.) Inc.: New York, NY, USA, 1958.
4. Haris, S.; Amdahl, J. Analysis of ship–ship collision damage accounting for bow and side deformation interaction. *Mar. Struct.* **2013**, *32*, 18–48. [CrossRef]
5. Törnqvist, R. *Design of Crashworthy Ship Structures*; Technical University of Denmark: Kongens Lyngby, Denmark, 2003.
6. Ehlers, S.; Broekhuijsen, J.; Alsos, H.S.; Biehl, F. Simulating the collision response of ship side structures: A failure criteria benchmark study. *Int. Shipbuild. Prog.* **2008**, *55*, 127–144. [CrossRef]
7. Storheimabe, M.; Amdahl, J.; Martens, I. On the accuracy of fracture estimation in collision analysis of ship and offshore structures. *Mar. Struct.* **2015**, *44*, 254–287. [CrossRef]
8. Driemeier, L.; Brünig, M.; Micheli, G.; Alves, M. Experiments on stress-triaxiality dependence of material behavior of aluminum alloys. *Mech. Mater.* **2010**, *42*, 207–217. [CrossRef]
9. Wang, Z.; Hu, Z.; Liu, K.; Chen, G. Application of a material model based on the Johnson-Cook and Gurson-Tvergaard-Needleman model in ship collision and grounding simulations. *Ocean Eng.* **2020**, *205*, 106768. [CrossRef]
10. Calle, M.A.G.; Alves, M. A review-analysis on material failure modeling in ship collision. *Ocean Eng.* **2015**, *106*, 20–38. [CrossRef]
11. Bai, Y.; Wierzbicki, T. Application of extended Mohr–Coulomb criterion to ductile fracture. *Int. J. Fract.* **2009**, *161*, 1–20. [CrossRef]
12. Lou, Y.; Huh, H. Evaluation of ductile fracture criteria in a general three-dimensional stress state considering the stress triaxiality and the lode parameter. *Acta Mech. Solida Sin.* **2013**, *26*, 642–658. [CrossRef]
13. Hu, Q.; Li, X.; Han, X.; Chen, J. A new shear and tension based ductile fracture criterion: Modeling and validation. *Eur. J. Mech. A/Solids* **2017**, *66*, 370–386. [CrossRef]
14. Bao, Y.; Wierzbicki, T. On fracture locus in the equivalent strain and stress triaxiality space. *Int. J. Mech. Sci.* **2004**, *46*, 81–98. [CrossRef]
15. Talebi-Ghadikolaee1, H.; Naeini1, H.; Mirnia, M.; Mirzai, M.; Alexandrov, S.; Gorji, H. Experimental and numerical investigation of failure during bending of AA6061 aluminum alloy sheet using the modified Mohr–Coulomb fracture criterion. *Int. J. Adv. Manuf. Technol.* **2019**, *105*, 5217–5237. [CrossRef]
16. Voormeeren, L.; Walters, C.; Tang, L.; Vredeveltdt, A. Estimation of failure parameters for finite element simulations based on a single state of stress and arbitrary stress-strain relation. In Proceedings of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering OMAE 2014, San Francisco, CA, USA, 8–13 June 2014.
17. Zhu, L.; Estefen, S.; Lourenço, M. Fracture criteria applied to numerical simulation of blowout preventer ram shearing. *Eng. Fail. Anal.* **2020**, *114*, 104596. [CrossRef]
18. Mohr, O. *Abhandlungen Aus Dem Gebiete Der Technischen Mechanik*; W. Ernst & Sohn: Berlin, Germany, 1914.
19. Bao, Y.; Wierzbicki, T. A comparative study on various ductile crack formation criteria. *J. Eng. Mater. Technol.* **2004**, *126*, 314–324. [CrossRef]
20. Yu, R.; Li, X.; Yue, Z.; Li, A.; Zhao, Z.; Wang, X.; Zhou, H.; Lu, T.J. Stress state sensitivity for plastic flow and ductile fracture of L907A low-alloy marine steel: From tension to shear. *Mater. Sci. Eng. A* **2022**, *835*, 142689. [CrossRef]
21. Dassault Systèmes. SIMULIA User Assistance 2020. Available online: https://help.3ds.com/2020/english/dssimulia_established/SIMACAESUBRefMap/simasub-c-vumat.htm?contextscope=all (accessed on 4 April 2024).
22. Ling, Y. Uniaxial true stress-strain after necking. *AMP J. Technol.* **1996**, *5*.
23. Liu, H.; Liu, K.; Wang, X.; Gao, Z.; Wang, J. On the Resistance of Cruciform Structures during Ship Collision and Grounding. *J. Mar. Sci. Eng.* **2023**, *11*, 459. [CrossRef]
24. Liu, K.; Zong, S.; Li, Y.; Wang, Z.; Hu, Z.; Wang, Z. Structural response of the U-type corrugated core sandwich panel used in ship structures under the lateral quasi-static compression load. *Mar. Struct.* **2022**, *84*, 103198. [CrossRef]
25. Liu, K.; Liu, H.; Li, Y.; Wang, Z.; Wang, J. A Simplified Method for Evaluating the Dynamic Response of the Metal Sandwich Structure under Explosion Load. *Metals* **2022**, *12*, 1933. [CrossRef]
26. Rice, J.R.; Tracey, D.M. On the ductile enlargement of voids in triaxial stress fields. *J. Mech. Phys. Solids* **1969**, *17*, 201–217. [CrossRef]
27. Cockcroft, M. Ductility and workability of metals. *J. Met.* **1968**, *96*, 2444.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Experimental Investigation of Ice Loads on Structures during Quasi-Static Compression Tests

Jinsong Xia ^{1,2}, Tongqiang Yu ^{3,*}, Kun Liu ³, Junji Xiang ³, Nan Zhao ^{1,2} and Junjie Liu ^{1,2}

¹ China Ship Scientific Research Center (CSSRC), Wuxi 214082, China; xjs@cssrc.com.cn (J.X.)

² National Key Laboratory of Ship Structural Safety, Wuxi 214082, China

³ School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang 212003, China

* Correspondence: justyu@163.com; Tel.: +86-182-6063-1192; Fax: +86-51184446543

Abstract: In polar ship hull structural designs, methods based on regulations are considered the most authoritative; however, they tend to be conservative and often exhibit a notable degree of redundancy. This study aims to evaluate the applicability of the empirical formula for ice load assessments by conducting a series of quasi-static indentation tests on scaled hull plates under laboratory-made ice blocks of different scales. The obtained data include ice loads, structural responses, and characteristics of ice pressure distribution. A detailed comparison of various formulas is provided, along with an examination of their differences and errors in comparison to experimental results. The objective of this paper is to offer technical support for ice load forecasting and assessment.

Keywords: ice load; quasi-static compression tests; scaled ship hull panels; empirical formulas

1. Introduction

The exploration and utilization of polar regions have attracted significant global attention, particularly due to the abundant fossil energy resources found in the Arctic region. Polar vessels inevitably encounter the impact of ice loads during navigation in icy regions, which involves a complex interaction process. It is widely acknowledged that the magnitude of ice loads depends on various factors such as the mechanical characteristics of the ice, the interaction mode between ice and structures, the geometric shape and dimensions of structures, the thickness and speed of ice blocks, and the failure mode of ice. Regarding the interaction mode between structures and ice, ice loads primarily result from processes involving compression, bending, and fracturing damage to ice floes, level ice, or icebergs. Notably, the compressive load imposed by ice is particularly significant and requires focused attention when assessing ship structure stability and safety.

The investigation of ice loads on bridges, piles, and structures has received considerable attention from scholars. Empirically based methods for assessing ice loads offer the advantages of convenience and efficiency. A widely accepted classical empirical formula was proposed by Korzhavin [1], who conducted extensive research on the characteristics of river ice loads based on measurements of dynamic and static pressures in Siberia. Similarly, Croasdale, Michel, Frederking, and other researchers employed similar on-site or experimental measurement methods [2–4], with variations in the types of ice studied, such as lake ice and freshwater ice. Currently, several classification societies and standard organizations, including Bureau Veritas (BV) [5] and the International Organization for Standardization (ISO) [6] recommend using empirically based methods for assessing ice loads. As methodologies continue to expand and improve, there is an active pursuit of organizational initiatives [7]. Masterson and Tibbo [8] conducted a comparative analysis of the different assumptions used in calculating ice loads based on measured data from Timco and Croasdale [9]. Kellner [10] provided a comprehensive review of current code-based methods for predicting ice loads. Barker [11] conducted experimental investigations on the

structural ice loads during the interaction between sea ice and cone structures. To this end, they employed seven model structures and analyzed the influence of ice speed, thickness, structural form, and stiffness on the ice loads acting on the models. Based on observing the process of ice breaking and the accumulation of broken ice, Li [12] measured the load and the size of broken ice and analyzed the influence of ship speed and ice thickness on the formation and accumulation of broken ice. Zong [13] used polypropylene wax to simulate floating ice and analyzed the characteristics of sea ice. With advancements in simulation technology, finite element calculations are increasingly being used for predicting and assessing ice loads, which serve as effective means for code-based verification [14–16]. Truong [17] proposed an ice pressure estimation method based on the numerical simulation stress response data influence coefficient method (ICM) and compared the estimated local ice pressure with the simulation results. The results show that the results are in good agreement under single dynamic loading cases, and the results are slightly deviated under dynamic loading combination cases. Based on the extended finite element method, Xu [18] verified the ice load obtained by the numerical simulation of the collision between ice and a rigid ship with the field test data. The results show that the numerical method is in good agreement with the ice load data obtained by the test. Zheng [19] used the ship–ice–water SPH method to analyze the predicted ice-breaking resistance under different conditions. The results show that the ice resistance is positively correlated with the bending strength and ice thickness, and the latter is the most important factor affecting the ice resistance. In summary, achieving a cohesive theoretical framework remains challenging when anticipating ice loads. Further research efforts require a stronger foundation of experimental data that aligns with the primary objective of this work.

The present study provides an introduction to the prevailing methodologies for static ice load prediction based on codes and empirical formulas. Additionally, quasi-static loading tests were conducted to investigate the interaction between local ship structures and the ice. This study compares the magnitude and variations of ice loads obtained from experimental measurements with theoretical calculations, thereby evaluating the efficacy and applicability of empirical methods for estimating ice loads. The purpose of this paper is to obtain a fast prediction method for ice load on flat structures under static ice extrusion. The relevant research results have positive significance for polar structural safety assessment and ice load prediction.

2. Quasi-Static Compression Experiments on Ice–Structure Interaction

2.1. Uniaxial Compression and Three-Point Bending Tests of Ice Specimens

The ice samples used in this study are polycrystalline ice made in the laboratory. This involved a process of freezing size-controlled crushed ice and degassed water. The requisite protocol for preparing these samples included an initial freezing phase in a controlled environment at $-20\text{ }^{\circ}\text{C}$ for 48 h, followed by a subsequent storage period exceeding two days in a defrost freezer before their utilization in the experiments. Compression and three-point bending tests (Support Span 60 mm) were conducted to acquire fundamental material parameters. The design of specimen dimensions adhered to the guidelines recommended by the International Association for Hydro-Environment Engineering and Research (IAHR) [20]. The compression specimens were sized at $70\text{ mm} \times 70\text{ mm} \times 175\text{ mm}$, while the bending specimens measured $75\text{ mm} \times 75\text{ mm} \times 700\text{ mm}$. The setup for these tests is illustrated in Figure 1.

During the tests, data on load and actuator displacement were recorded, and the mechanical parameters were computed accordingly, as illustrated in Figure 2. The calculated bending and compression strengths of the ice samples are presented in Tables 1 and 2. The mechanical properties assessment, as well as the subsequent quasi-static extrusion test of the structure–ice interface, were conducted under controlled laboratory conditions, with an ambient temperature of $25\text{ }^{\circ}\text{C}$, and the installation preparation time was 3 min. The loading speed was set at 1 mm/s .

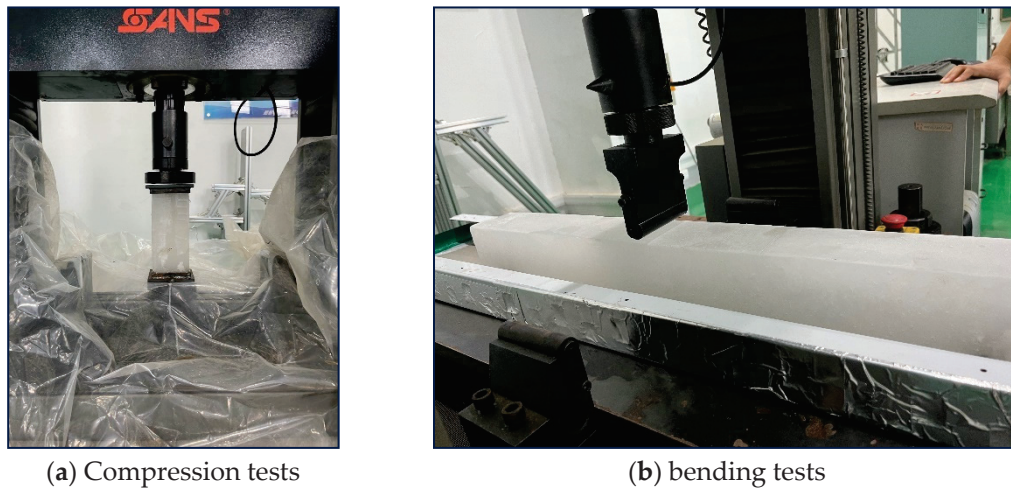


Figure 1. Compression and bending test setup for ice specimens.

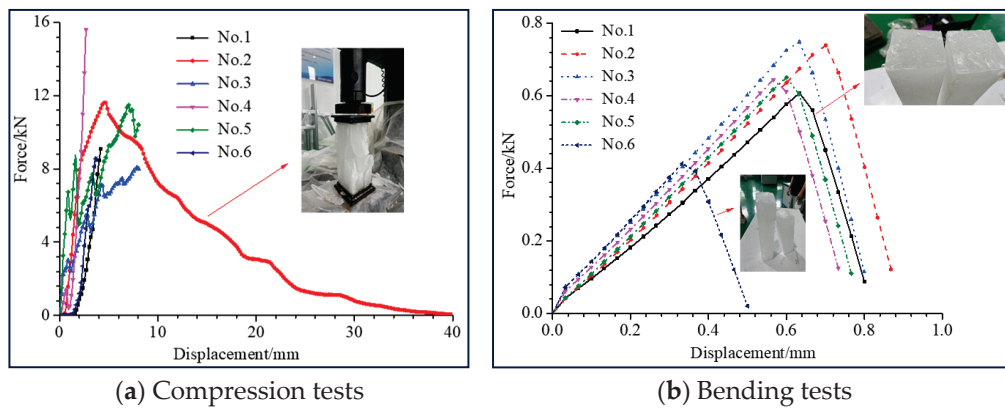


Figure 2. Load–displacement curve of the ice sample compression and bending tests.

Table 1. Calculated compressive strength of ice samples.

Test Number	Ambient Temperature (°C)	Compressive Strength (MPa)	Mean Compressive Strength (MPa)
No.1	25	1.86	2.20
No.2	25	2.37	
No.3	25	1.65	
No.4	25	3.19	
No.5	25	2.35	
No.6	25	1.77	

Table 2. Calculated bending strength of ice samples.

Test Number	Ambient Temperature (°C)	Bending Strength (MPa)	Mean Bending Strength (MPa)
No.1	25	1.59	1.66
No.2	25	1.93	
No.3	25	1.96	
No.4	25	1.69	
No.5	25	1.71	
No.6	25	1.02	

The compression and bending strengths of the ice samples obtained by freezing crushed ice were found to be within the range of typical strength values for natural existing ice (typically, 0.5 MPa~5 MPa for uniaxial compressive strength and 1 MPa~2 MPa for bending strength) [21]. Moreover, it is evident that the experimental results demonstrate good stability. As shown in Figure 2, the results of six compression tests of ice show similar brittle failure. The load–displacement curves show a multi-peak shape. The six bending tests of the ice body are also brittle failures, and the fracture position is slightly different. The load peaks and curves of compression and bending tests show certain differences because the ice samples prepared in the laboratory cannot be standardized. It should be noted that the variability observed in bending test No. 6 can be attributed to inherent defects in the test specimen itself.

2.2. The Experimental Setup and Its Implementation

The primary objective of the model test is to investigate the underlying mechanism during the ice-breaking process and evaluate methodologies for calculating ice loads. The stiffened panel, as illustrated in Figure 3, utilized in the experiments was scaled down from a specific ship's partial structure, with its design dimensions depicted in Figure 3a. Comprising panels, reinforcing members, and girders are all constructed from mild steel (Q235), and the stiffened panel consists of three distinct types of components.

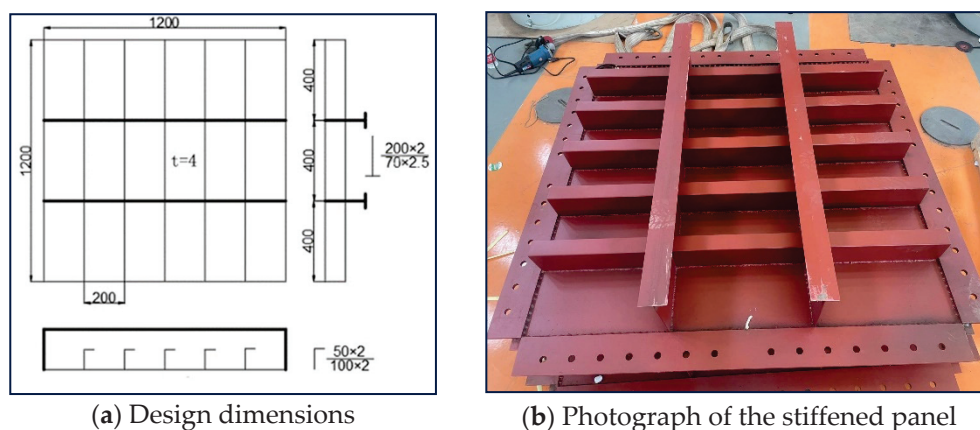


Figure 3. The stiffened panel was used in the experiment.

In the experimental setup, the ice block specimens are classified into three distinct types based on their respective contact areas: specifically, 300 mm × 75 mm, 300 mm × 150 mm, and 300 mm × 225 mm. All of the ice block specimens have a height of 600 mm, a top-end width of 300 mm, and a thickness of 300 mm, as depicted in Figure 4.

The quasi-static compression tests were performed using a 10-ton electro-hydraulic servo system, as illustrated in Figure 5a. A set of meticulously designed fixtures was developed and utilized in the experiments, as depicted in Figure 5b. The connecting joint acts as an intermediary between the ice block and the actuator. The upper section of the joint is securely fastened to the actuator using bolts, while the lower section is integrated with the ice block through freezing with hemp ropes. The stiffened panel is bolted to the support fixtures, which are specifically designed to provide ample rigidity, ensuring that there is no deformation within the frame's boundaries. It should be noted that during the experiment, the longer side of the ice block's lower part was aligned parallel to the direction of the stiffeners, while its shorter side was perpendicular to them. The actuator's speed was set at 1 mm/s, and load and displacement data were systematically recorded throughout each process. Three replicated experiments have been conducted for each type of ice block.

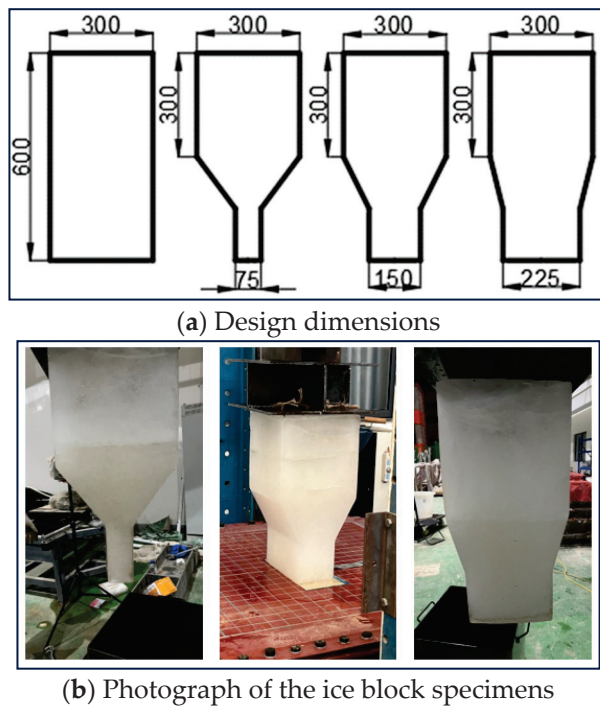


Figure 4. The ice block specimens were used in the experiment.

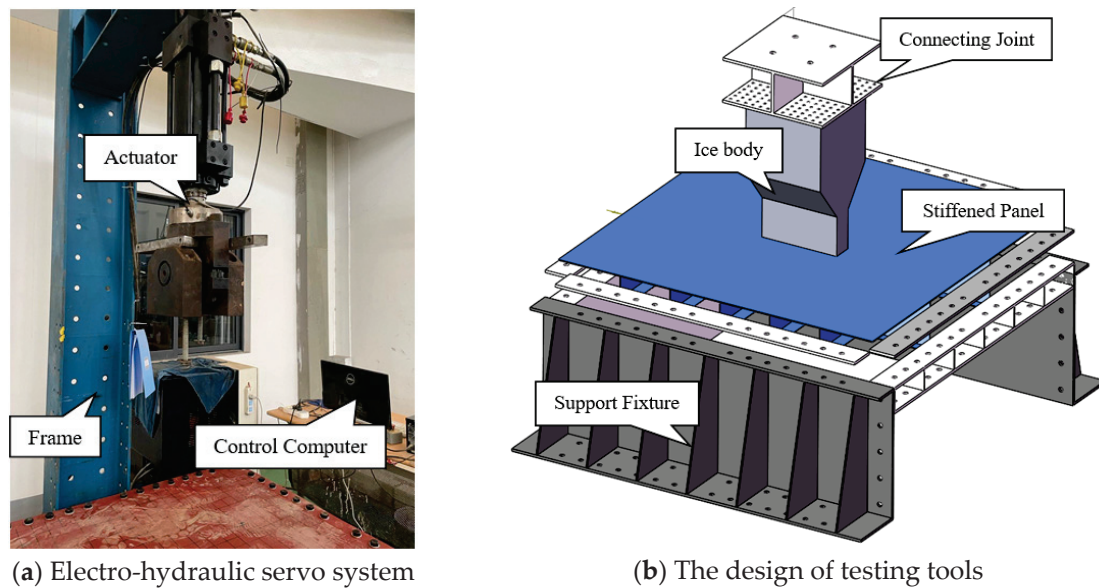


Figure 5. Test loading system.

2.3. Experimental Results

2.3.1. Analysis of Ice Failure Mode

The quasi-static compressive experiment in Figure 6 presents an image sequence depicting the interaction process between the ice block and stiffened panel with varying contact sizes. It is evident that regardless of the contact size, significant cracking consistently occurs at the middle stiffener of the panel (indicated by red arrows) due to the enhanced strength at these locations, rendering the ice block incapable of sustaining applied shear loads. When subjected to contact sizes of 300 mm $\times$ 75 mm and 300 mm $\times$ 150 mm, spalling occurs as a result of major cracking damage in the ice; however, a substantial load-bearing capacity is maintained by the remaining ice block. In contrast, when using a contact surface

size of 300 mm × 225 mm, direct crushing of the ice takes place, leading to destruction characterized by pronounced shattering.

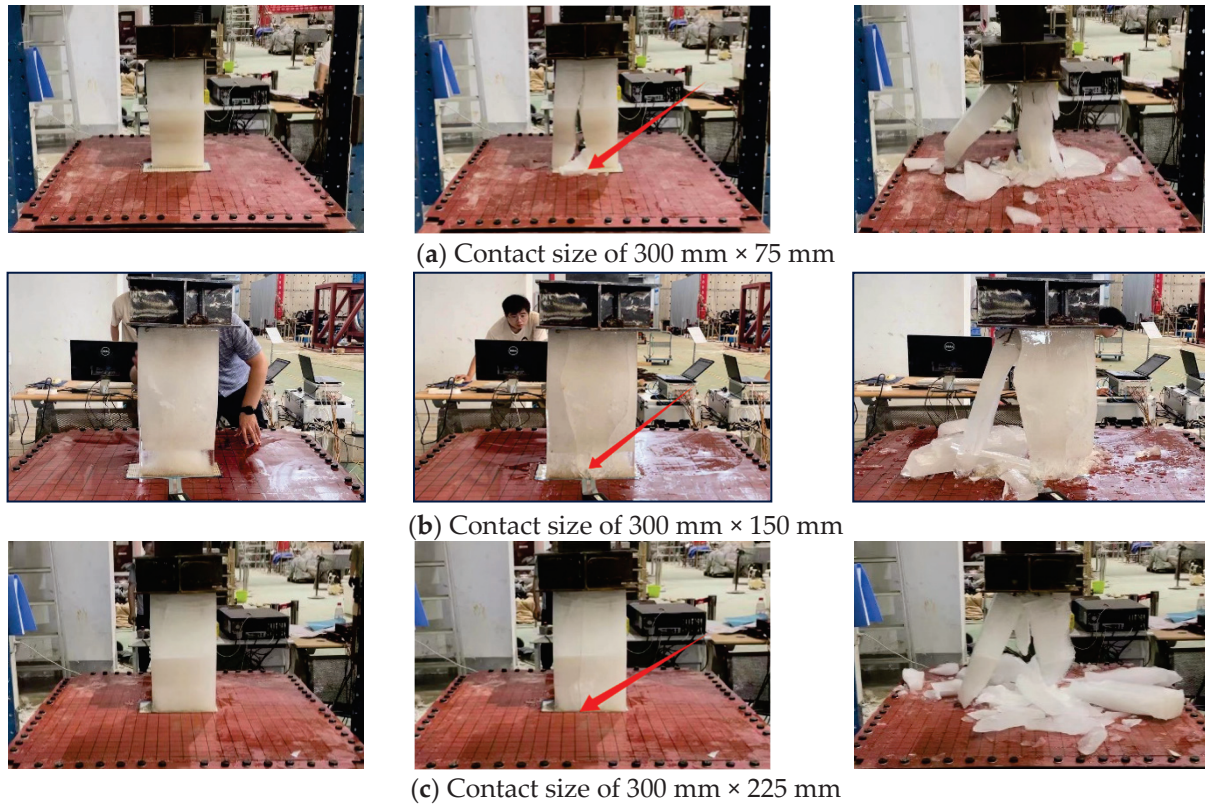


Figure 6. Ice destruction process with different contact sizes.

2.3.2. Structural Deformation

No significant or easily observable plastic deformation was observed in the stiffened panel in all test cases. Only when the contact area is relatively large does a minor indentation deformation occur, as depicted in Figure 7. The maximum deformation measured was approximately 6 mm. Due to the presence of a middle stiffener, the deformation is evenly distributed on both sides of it and remains within a relatively narrow range. Based on photographic evidence from the experimental procedure, it can be inferred that the panel's deformation primarily stems from residual ice blockage following major cracking. This observation suggests that even after fracture, the ice block retains considerable load-bearing capacity.

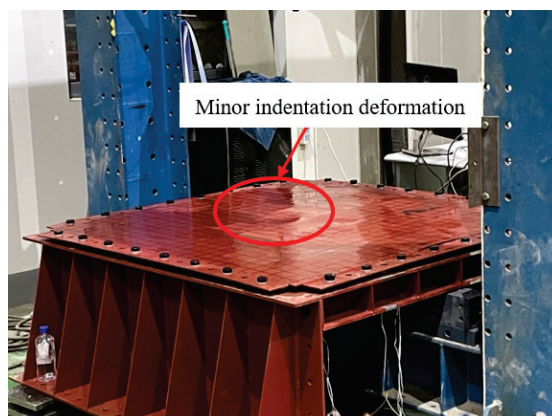


Figure 7. Deformation of the stiffened panel (contact area: 300 mm × 225 mm).

2.3.3. Load–Displacement Data during the Test Process

Figure 8 presents the load–displacement curves under different contact sizes. It can be observed that during the compression process, the load curves exhibit characteristics of slow loading and rapid unloading in all cases. The load–displacement curve changes effectively reflect the ice block’s failure process. When the contact size is $300\text{ mm} \times 75\text{ mm}$, the load curve exhibits typical non-linear characteristics with multiple peaks. The maximum peak load of the first peak load can reach 29.04 kN , and the minimum is 18.83 kN . The failure process of the ice in the three repeated tests reflects the random destructiveness of the ice, and similar load variation characteristics are observed. The first peak values are 20.72 kN , 18.83 kN , and 29.04 kN , respectively. In addition, it can be clearly seen that the maximum peak load in the test is significantly greater than the first peak (45.79 kN , 29.63 kN , and 61.61 kN), indicating that the bearing capacity of residual ice is indeed great. When the contact size is $300\text{ mm} \times 150\text{ mm}$, the peak number is significantly reduced, the displacement value is significantly reduced, and the uniformity is stable at about 20 mm , and the repeated test load characteristics also reflect similarity. The maximum peak loads recorded were 45.20 kN , 46.46 kN , and 30.17 kN , respectively. When the contact size is $300\text{ mm} \times 225\text{ mm}$, the load–displacement curve exhibits only a single peak, with the peak values being 84.54 kN , 189.47 kN , and 171.6 kN , respectively. This result stems from the fact that the contact area between the stiffened plates and the ice bodies of this series is the greatest compared to the other two series. Furthermore, following the vertical cracking failure of the ice body at the rib, no ice body suffered a fracture or detachment during the subsequent loading process. The ice body continued to withstand loading until it surpassed its maximal bearing capacity, at which point it fractured and ceased contact with the plate. Under this condition, the ice body shows brittle failure. The reason for the smaller load in the first test is still due to the sliding of the smaller ice blocks after cracking, as shown in Figure 8(c3).

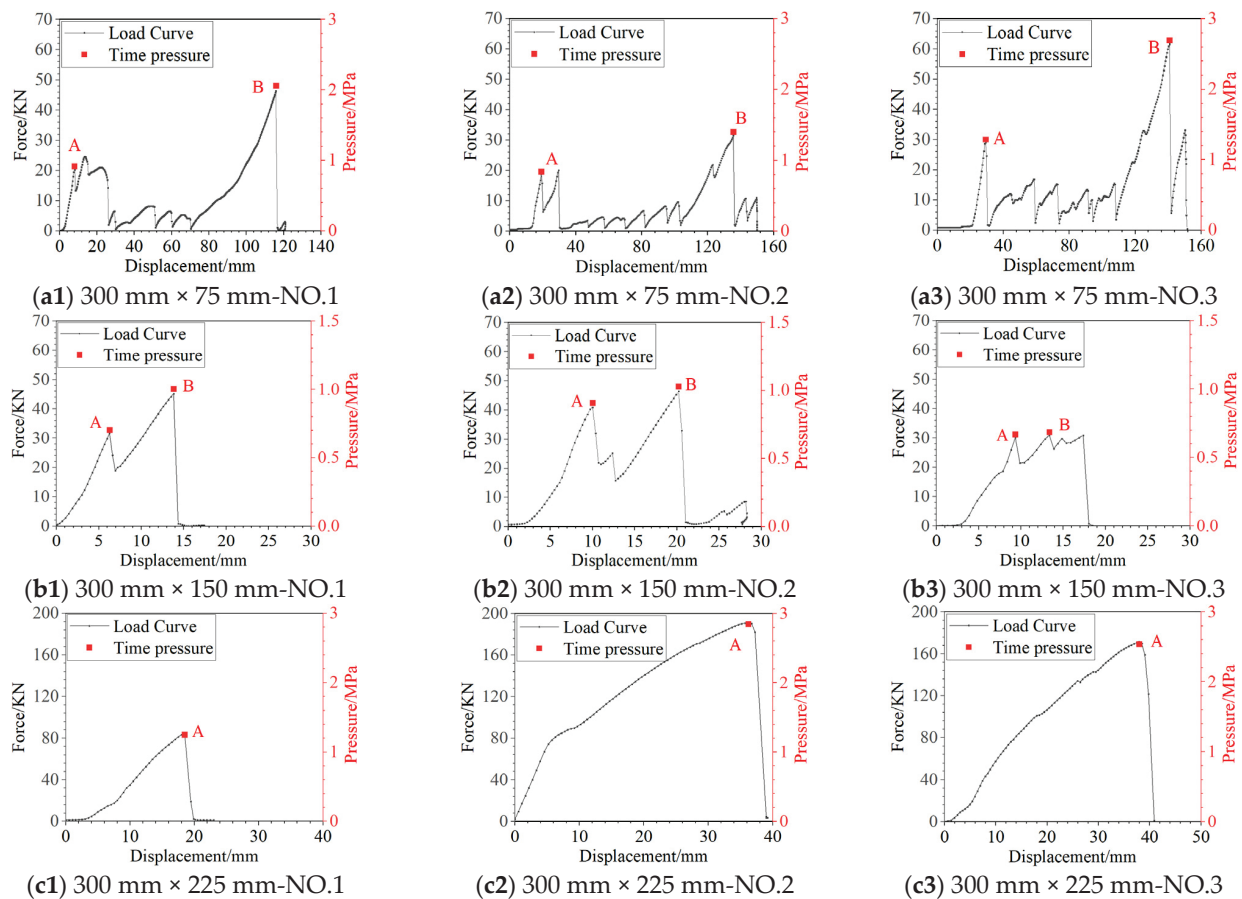


Figure 8. Load–displacement curves under various contact sizes.

Point A and point B represent the pressure values corresponding to the first peak load and the maximum peak load moment, respectively. These values are calculated by the moment load and the initial contact area, as shown in Figure 8. It is worth noting that when the contact area is 300 mm × 225 mm and the load curve only shows one peak, the first peak load value is equal to the maximum load value. Further, Table 3 records the pressure data under different working conditions in detail. Through the analysis of these data, we can observe that the overall trend of the pressure value increases with an increase in the contact area. However, there is an interesting phenomenon that when the contact area is 300 mm × 150 mm, the corresponding pressure value is lower than that when the contact area is 300 mm × 75 mm. After in-depth analysis, we found that this is mainly because during the test, the first peak load and the maximum load measured when the contact area is 300 mm × 150 mm are almost the same as those when the contact area is 300 mm × 75 mm. However, because the contact area of the former is twice that of the latter, this abnormal pressure value occurs.

Table 3. Load and pressure values under different sizes.

Case Number	First Peak Load (KN)	First Peak Pressure (MKN)	Maximum Load (KN)	Maximum Load Pressure (MKN)
300 mm × 75 mm-NO.1	20.72	0.92	45.79	2.06
300 mm × 75 mm-NO.2	18.83	0.84	29.63	1.40
300 mm × 75 mm-NO.3	29.04	1.20	29.04	2.70
300 mm × 150 mm-NO.1	31.68	0.70	45.20	1.00
300 mm × 150 mm-NO.2	40.97	0.91	46.46	1.03
300 mm × 150 mm-NO.3	30.17	0.67	30.94	0.69
300 mm × 225 mm-NO.1	84.54	1.25	84.54	-
300 mm × 225 mm-NO.2	190.45	2.85	190.45	-
300 mm × 225 mm-NO.3	171.60	2.54	171.60	-

The first peak load along with the maximum load values under different cases are summarized in Table 3. It is observed that the maximum load values demonstrate a nonlinear positive correlation with the contact area. Additionally, it should be noted that, for some of the working conditions, the initial peak load is identical to the maximum peak load.

3. Static Ice Load Calculation Method

Currently, ice load calculation standards have been implemented in countries such as the United States and Canada, with numerous scholars developing their empirical formulas based on experimental findings. It is important to note that, for the safety of Arctic navigation, a significant portion of ice load research primarily focuses on sea ice phenomena like icebergs and level ice. However, countries with extensive inland regions and cold climates, such as the former Soviet Union, also attach great importance to studying ice loads in inland rivers. Nevertheless, the fundamental principle of calculating ice loads remains straightforward, that is, as follows:

$$F = \sigma_{\max} A \quad (1)$$

Herein, F , $\sigma_{\max}$, and A respective variables represent the calculated ice force, the compressive strength of the ice, and the area of mutual contact. This formula holds for static conditions with a flat contact surface and the absence of friction. However, considering practical application scenarios, it becomes imperative to take into account factors such as the projected area component on the vertical cross-section of the contact surface and parameters related to slippage. Consequently, a revision is necessary to develop an empirical calculation method. Currently, there are various empirical formulas available

for calculating static ice force, which can be broadly classified into categories based on linear functions and those founded on exponential functions.

3.1. The Method of Predicting Static Ice Loads Based on Linear Formulas

The magnitude of ice loads is influenced by multiple factors, and the empirical formulas are typically characterized by the quantitative representation of various parameters. In their initial stages, these formulas were commonly linear, with both single and multiple coefficients to facilitate computational simplicity, as demonstrated in Equation (2).

$$F = \alpha D h \sigma_c \quad (2)$$

In the formula, D represents the projected width of the structure's ice-facing surface, h denotes the thickness of the ice, σ_c is indicative of the uniaxial compressive strength of the ice, and α accounts for comprehensive correction factors that consider modifications to the actual stress state caused by variables such as ice velocity, contact conditions, and local compression. These formulas incorporate experimental findings and theoretical approaches, possessing explicit physical significance while maintaining dimensional consistency and broad applicability. Based on linear formulas, static ice loads primarily rely on both the uniaxial compressive strength of the ice and the contact area between it and the structure. Table 4 provides a summary of similar formulas [22].

Table 4. The calculation formula for static ice load is based on the linear formula.

Name	Expression	Meaning of the Parameters	Background
Korzhavin	$F = I(V/V_0)^{-1/3} m K D H \sigma_c$	I : local compression coefficient; v : relative velocity; v_0 : reference velocity; m : structural shape coefficient; K : contact condition coefficient; H : ice thickness.	Based on a large number of experimental data and theoretical analysis of Siberian rivers.
Afanasev (1971)	$F = I m D h \sigma_c, \quad I = \begin{cases} (1 + 5 \times H/D)^{1/2} & (1 < D/H \leq 6) \\ 2.5 & (D/H = 6) \\ 4.17 - 1.67 D/H & (0.1 < D/H \leq 1) \\ 4.0 & (D/H = 1) \end{cases}$	I : local compression coefficient; m : structural shape coefficient.	Based on Korzhavin, further research is carried out, and the calculation method under different D/H ratios is obtained.
Lu	For vertical piers: $F = a C D H \sigma_c$ For inclined piers, horizontal force: $F_H = a_w H^2 \tan \beta$ Vertical force: $F_V = a_w^\sigma H^2$	a : regional coefficient; C : comprehensive correction coefficient; σ_w : ice's flexural strength; β : the angle of inclination of the ice-breaking edge of the bridge pier.	This paper is based on a large number of experiments on the ice pressure on the No.3 pier of the Songhua River Bridge caused by the ice flow in the Hal Glacier section of the Songhua River in China.
Former Soviet Union's standards	$F = I m D h \sigma_c$	I : local compression coefficient.	Based on the Arctic coastline scene of the former Soviet Union, the ice pressure acting on the vertical pier is analyzed.
Russian Ice Load Regulations	$F = \varphi_i D H \sigma_c$	φ_i : structural ice-facing surface shape coefficient; H : ice thickness.	The ice surface of the bridge pier is vertical, and the ice load on the bridge pier is applied.
American API2A Formula	$F = C D H \sigma_c$	C : coefficient dependent on the structural shape, ice velocity, and other factors.	For the flowing ice force acting on a single column, the larger the ratio of pile diameter to ice thickness, the smaller the coefficient ' c '.
American APIRP2N Formula	$F = I m K D h \sigma_c$	I : local compression coefficient; K : contact coefficient.	Based on the ice flow force on a single column in the Cook Bay of Alaska, USA.

Table 4. Cont.

Name	Expression	Meaning of the Parameters	Background
Canadian CAS Formula (1974)	$F = DhP_c$	P_c : effective ice pressure.	Based on the unique ice environment and cold temperate climate in Canada, the force of flowing ice on buildings is proposed.
Canadian Standards for Bridge Piers	$F = C_n DH\sigma_c$	C_n : inclination coefficient of bridge pier fore-edges.	Based on different inclination angles of the leading edge of the pier, there are different inclination coefficients.
Canadian Lighthouse Formula (1975)	$F = mDh\sigma_c$	m : comprehensive correction coefficient.	Ensure that the Canadian offshore lighthouse structure can withstand the ice load that may be suffered under extreme conditions.
Chinese Offshore Platform Standards	$F = mK_1K_2Dh\sigma_c$	K_1 : local compression coefficient; K_2 : contact coefficient; σ_c : ice's compressive strength.	Under the action of wind and current, the ice load of a large area of the ice sheet on the vertically isolated pile.

3.2. Exponential Formula-Based Predictive Method for Static Ice Loads

The expression method for static ice loads, established by Schwarz through regression analysis of experimental data from model experiments, is based on an exponential formula and can be articulated as follows:

$$F = \beta D^{0.5} H^\gamma \sigma_c \quad (3)$$

In the formula, β represents a correction coefficient that is typically a constant value under particular environmental circumstances.

This type of formula represents an advancement in the existing methods for calculating ice loads, which were initially developed based on linear formulas and subsequently refined by incorporating various exponential terms derived from extensive experimental data. It is specifically tailored to suit particular environmental conditions, with a relatively limited scope of application and involving parameters that lack dimensional consistency. However, it offers highly targeted results with enhanced precision. Such formulas are presented in Table 5 [22]. This category of formulas combines and simplifies the influence of different factors from the first category of formulas on the calculation of ice forces, thereby providing unique practical value for real-world applications.

Table 5. Calculation formula of static ice load based on the exponential formula.

Name	Expression	Notes	Background
Schwarz Formula	$F = 3.75D^{0.5}H^{1.1}\sigma_c$	F : unit in KN; D : unit in cm; σ_c : unit in KN/cm ² .	A large number of experimental and theoretical studies have been conducted in the Baltic Sea, North Sea, and North Atlantic, and a multi-coefficient formula is proposed.
Hamayaka Formula	$F = CD^{0.5}H\sigma_c$	C : correction coefficient. For circular cross-sections, it takes 5.0; for rectangular cross-sections, it takes 6.8.	Based on the ice situation in the Beibu Gulf of Japan.

4. Comparison of Ice Load Magnitudes from Experiments and Empirical Formulas

The present study meticulously compiles empirical methodologies for estimating ice loads on marine piles and bridge piers. Given the analogous loading patterns observed in side shell structures with significant stiffness, the extrapolation of these empirical formulas

to such constructions is considered justifiable. By disregarding the progressive disintegration of the ice block, we adopt the initial peak value on the load–displacement curve as the reference experimental load, facilitating a comparative analysis with outcomes derived from various empirical formulas, as depicted in Figure 9. It is evident that results obtained from the same empirical formula positively correlate with the contact area. Due to the conservative parameters and assumptions employed in standards and empirical formulas, the resultant load values generated in practice are comparatively higher. Notably, when the aspect ratio of the contact area is smaller, discrepancies in load estimations predicted by these methods become more pronounced. Additionally, it can be observed that the calculation results from the APIRP2N formula consistently overestimate. This can primarily be attributed to contact coefficients being derived from ice conditions in Cook Inlet, Alaska, USA, where parameter values are significantly higher than those used in other standards.

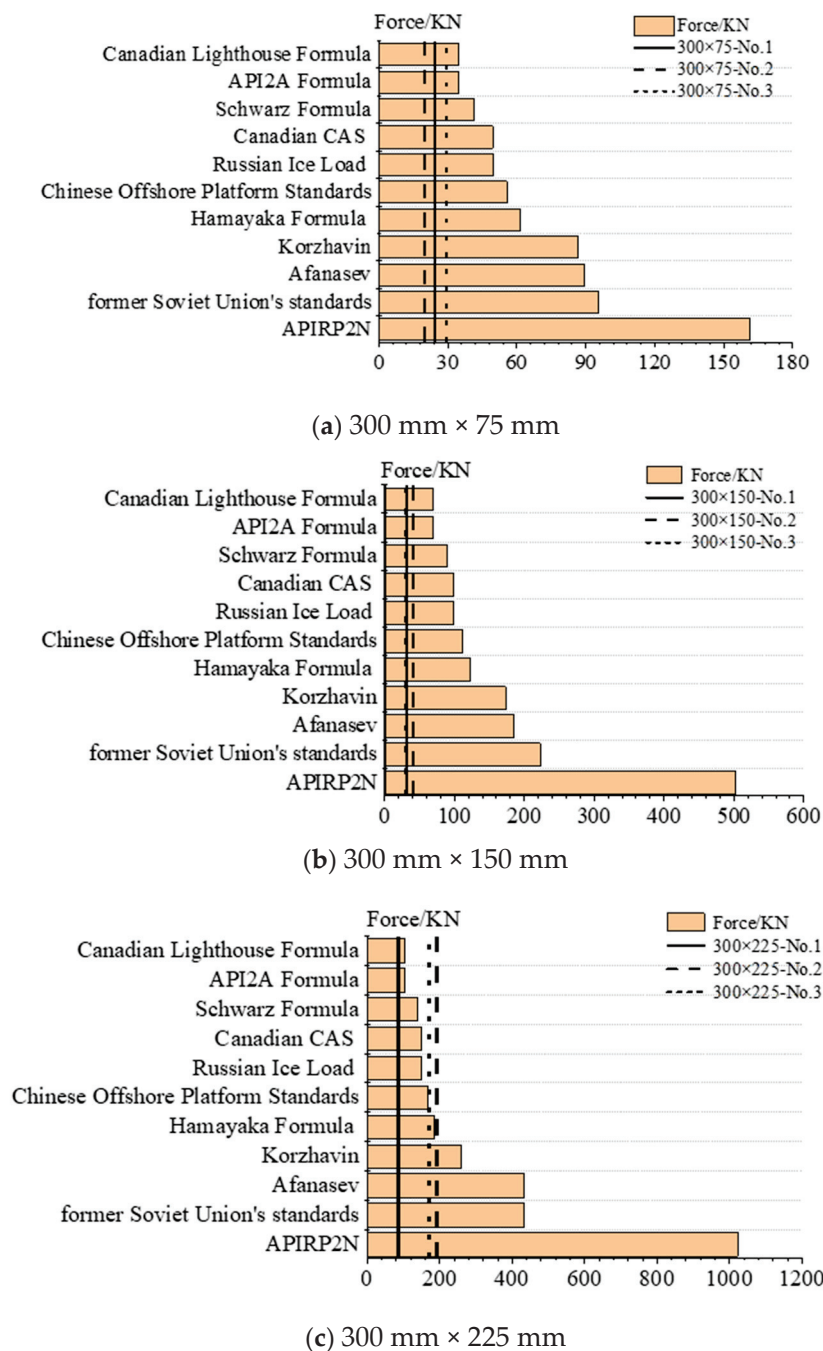


Figure 9. Comparison of results from various empirical formulas with actual measurement data.

5. Discussion

5.1. Comparative Analysis of Different Empirical Formulas

5.1.1. Identifying the Factors for Determining Correction Coefficient

The correction coefficient, as mentioned previously, is an empirical value obtained by fitting a large set of experimental data. It may be a composite correction factor or consist of multiple sub-factors to collectively account for the influence of various factors. The determination of this coefficient should consider the following considerations:

(1) Contact state between ice and structure

In both categories of formulas (Equations (2) and (3)), the compressive strength σ_C is determined through uniaxial compression tests. However, it should be noted that these tests do not consider the three-dimensional stress state of ice, which significantly enhances its strength under confining pressure. This distinction is crucial in understanding different scenarios, such as ship collisions with ice flies or icebergs and the compression of river ice against bridge piers. Therefore, when dealing with substantial ice thicknesses, it becomes necessary to further calibrate the correction coefficient to account for the enhanced strength beyond what is determined by uniaxial compression.

(2) Ice–structure contact conditions

In ice force calculation formulas, ' Dh ' (Dh in Equation (2) and $D^i H^j$ in Equation (3)) represents the projected contact area between ice and the structure. However, due to the brittle nature of ice, the spalling of ice fragments results in numerous irregular notches, thereby reducing the effective contact area. The faster the compression rate of ice, the more pronounced this effect becomes. Moreover, it is unlikely for the entire ice mass to attain its ultimate strength and fail uniformly across the actual contact area, leading to disparities between theoretical and practical contact areas. It should be noted that strain rate and interaction velocity are closely intertwined; however, in real-world scenarios, sea ice velocity near structural entities exhibits significant variability. Consequently, adjusting parameters α and β may not yield tangible engineering benefits in such contexts. Therefore, only the Korzhavin formula incorporates both strain rate and uniaxial compressive strength.

5.1.2. Ice Load Magnitude and Contact Surface Shape Characteristics

In the second category of formulas, the formulas are obtained by applying an exponential transformation to the length and width dimensions of the contact surface in the first category. Consequently, the influencing factors to be considered are similar, among which the most critical is the shape and angle of the contact surface.

(1) Structural shape coefficient

The shape factor ' m ' represents the influence of the structural shape on ice forces and is generally determined based on the shape of the structure's ice-facing surface. Typically, a value of 1 is commonly assigned for flat contact surfaces, while cylindrical (or semi-circular) surfaces are often given a value of 0.9. For angular shapes, which are defined by the vertex angle ' 2α ', the corresponding value can be obtained from Table 6 [23].

Table 6. Shape coefficient values for different angles.

2α	45°	60°	75°	90°	120°	180°
m	0.6	0.65	0.69	0.73	0.81	1.0

In certain empirical formulas (see Table 1), the impact of the inclination coefficient on ice loads needs to be considered. Table 7 presents the ratios of numerical force to horizontal force under varying inclination coefficients (angles) [24]. It is noteworthy that at inclination angles exceeding a certain threshold (commonly 30°), the mode of ice failure may transition from compressive to bending failure. Consequently, in such scenarios, the calculation of ice pressure should employ the bending strength of the ice, necessitating corresponding modifications to the computational formulas.

Table 7. The ratio of vertical to horizontal forces under various inclination coefficients.

Inclination Angle		Inclination Coefficient C_n	The Ratio of Vertical to Horizontal Forces FV/FH
α	β		
0	90	1.00	0
15	75	0.75	0.11
20	70	0.75	0.20
25	65	0.75	0.29
30	60	0.75	0.39
35	55	0.50	0.50
40	50	0.50	0.61
45	45	0.50	0.74

5.2. Modification of the Korzhavin Formula Based on Experimental Outcomes

The Korzhavin formula, as a classic method for static ice load prediction, encompasses a comprehensive range of parameters. Additionally, based on the comparative results presented earlier, the Korzhavin formula demonstrates superior applicability and accuracy relative to other empirical formulas. The loading conditions of this experiment comply with the requirements of the Korzhavin formula, thereby justifying its use for calculating ice loads. Therefore, attention must be focused on the selection of the local compression coefficient ' T '. The literature indicates that when the width of the ice floe exceeds 15 times the pile diameter, ' T ' should be set at 2.5. However, this method of determination is evidently inapplicable in the context of this study. To enhance the alignment of the Korzhavin formula with the specific research object, it is imperative to amend the localized compression coefficient incorporated in the formula that is not consistent with the authentic research conditions. Therefore, we propose a fresh interpretation of the compression coefficient based on the test environment and results. Such a modification will ensure greater accuracy in our investigations. A compression coefficient deduced from experimental results is presented in Table 8.

Table 8. Comparison of results from improved Korzhavin formula and experiments.

Case Number	First Peak Load (KN)	Local Compression Coefficient Value	Average Compression Coefficient	COV (Coefficient of Variation)	Empirical Formula-Derived Ice Loads (KN)	Relative Error (%)
300 mm × 75 mm-NO.1	20.72	0.71	0.70	0.7/2.5 = 0.28	24.15	1.79
300 mm × 75 mm-NO.2	18.83	0.54				28.25
300 mm × 75 mm-NO.3	29.04	0.84				16.84
300 mm × 150 mm-NO.1	31.68	0.42	0.49	0.49/2.5 = 0.196	33.90	7.01
300 mm × 150 mm-NO.2	40.97	0.46				17.26
300 mm × 150 mm-NO.3	30.17	0.59				12.36
300 mm × 225 mm-NO.1	84.54	0.61	1.07	1.07/2.5 = 0.428	111.23	31.57
300 mm × 225 mm-NO.2	190.45	1.37				41.59
300 mm × 225 mm-NO.3	171.60	1.24				35.18

It is observable that the localized compression coefficients derived from experimental results are consistently lower than 2.5. If a discrepancy of 30% is considered acceptable, then, with the exception of certain cases, the overall error margin between the ice load calculated using empirical formulas and that obtained from experiments generally falls within an acceptable range, with the minimum error being 1.79%. This observation substantiates the rationality of the employed empirical formula.

6. Conclusions

This study focuses on the prediction of quasi-static ice loads, employing a comprehensive review of empirical formula-based forecasting methods and validation through quasi-static compression tests of ice–structure interaction. An assessment of the accuracy and applicability of various empirical calculation methods was conducted. The conclusions are as follows:

- (1) Various standards and empirical formulas exist for static ice load prediction, which can be classified into two main types based on their formulation: linear and exponential. In practical applications, the results obtained from different empirical formulas often exhibit significant variation due to variations in engineering contexts, ice conditions, etc.
- (2) Quasi-static compression tests of ice–structure interaction were conducted. The stiffened panel did not undergo any noticeable or easily observable plastic deformation. In all the tests studied in this paper, the ice block consistently exhibits significant cracking at the middle stiffener of the panel, and the maximum load values demonstrate a nonlinear positive correlation with the contact area.
- (3) A comprehensive discourse on the determination of various parameters was undertaken, encompassing aspects such as the correction coefficient, characteristics of the contact surface shape, and the structural shape coefficient. The Korzhavin formula is shown to exhibit enhanced applicability and precision in comparison to other empirical formulas. Therefore, a compression coefficient deduced from experimental results is presented, which also substantiates the empirical formula’s rationality and efficacy.

Using the model test method, the ice load size and load change can be directly obtained and further compared with the calculated values of the empirical formula. However, due to certain error limitations of the test technology, it is impossible to accurately simulate and restore the real ice-breaking scene. In addition, the factors that affect the accuracy of the empirical formula will be further studied, and the test technology will be continuously improved in the future.

Author Contributions: Formal analysis, J.X. (Jinsong Xia); methodology, J.X. (Jinsong Xia); project administration, T.Y. and N.Z.; resources, K.L.; supervision, K.L., and J.L.; validation, J.X. (Junji Xiang) and T.Y.; writing—original draft, J.X. (Junji Xiang) and J.X. (Jinsong Xia); writing—review and editing, J.X. (Junji Xiang). All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by a Project supported by the National Natural Science Foundation of China grant number 52192694, and National Key Technologies Research and Development Program grant number 2022YFB3306200.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Korzhavin, K.N. *Action of Ice on Engineering Structures*; Defense Technical Information Center: Fort Belvoir, VA, USA, 1962.
2. Croasdale, K.R.; Morgenstern, N.R.; Nuttall, J.B. Indentation Tests to Investigate Ice Pressures on Vertical Piers. *J. Glaciol.* **1977**, *19*, 301–312. [CrossRef]
3. Michel, B.; Toussaint, N. Mechanisms and Theory of Indentation of Ice Plates. *J. Glaciol.* **1977**, *19*, 285–300. [CrossRef]
4. Frederking, R.; Timco, G.; Reid, S. *Sea Ice Floe Impact Experiments*; National Research Council of Canada: Quebec, QC, Canada, 1999. [CrossRef]
5. Veritas, B. NI565 Ice Characteristics and Ice/Structure Interactions. Guidance Note NI, 2010. Available online: <https://marine-offshore.bureauveritas.com/ni565-ice-characteristics-and-icestructure-interactions> (accessed on 28 March 2024).
6. *ISO19906:2019(E)*; Petroleum and Natural Gas Industries—Arctic Offshore Structures. ISO: Geneva, Switzerland, 2019.
7. Masterson, D.M.; Kouzmitchev, K.; deWaal, J. Russian SNiP 2.06.04-82 and Western Global Ice Pressures—A Comparison. In Proceedings of the 17th International Conference on Port and Ocean Engineering under Arctic Conditions, Trondheim, Norway, 16–19 June 2003.

8. Masterson, D.M.; Susan Tibbo, J. Comparison of Ice Load Calculations Using ISO 19906, CSA, API and SNiP. In Proceedings of the 21st International Conference on Port and Ocean Engineering under Arctic Conditions (POAC'11), Montréal, QC, Canada, 10–14 July 2011; Volume 2.
9. Timco, G.; Croasdale, K. How Well Can We Predict Ice Loads? *IAHR* **2006**, *1*, 167–174.
10. Kellner, L.; Herrnring, H.; Ring, M. Review of Ice Load Standards and Comparison with Measurements. In Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering–OMAE, American Society of Mechanical Engineers, Trondheim, Norway, 25–30 June 2017; Volume 57762, p. V008T07A028.
11. Gravesen, H.; Sørensen, S.L.; Vølund, P.; Barker, A.; Timco, G. Ice Loading on Danish Wind Turbines: Part 2. Analyses of Dynamic Model Test Results. *Cold Reg. Sci. Technol.* **2005**, *41*, 25–47. [CrossRef]
12. Zhou, L.; Riska, K.; von Bock und Polach, R.; Moan, T.; Su, B. Experiments on Level Ice Loading on an Icebreaking Tanker with Different Ice Drift Angles. *Cold Reg. Sci. Technol.* **2013**, *85*, 79–93. [CrossRef]
13. Zong, Z.; Yang, B.Y.; Sun, Z.; Zhang, G.Y. Experimental Study of Ship Resistance in Artificial Ice Floes. *Cold Reg. Sci. Technol.* **2020**, *176*, 103102. [CrossRef]
14. Su, B. Numerical Predictions of Global and Local Ice Loads on Ships. Ph.D. Thesis, Norwegian University of Science and Technology, Trondheim, Norway, 2011.
15. Liu, Z.; Amdahl, J.; Løset, S. Numerical Simulation of Collisions between Ships and Icebergs. In Proceedings of the International Conference on Port and Ocean Engineering under Arctic Conditions, POAC, Lulea, Sweden, 9–12 June 2009; Volume 1.
16. Yu, T.; Liu, K.; Wang, G.; Liu, J.; Wang, Z. A Tri-Axial Ice Model for Simulating Ice-Stiffened Panel Impact: Experiments and Numerical Modeling. *Mar. Struct.* **2023**, *88*, 103358. [CrossRef]
17. Truong, D.D.; Jang, B.-S. Estimation of Ice Loads on Offshore Structures Using Simulations of Level Ice-Structure Collisions with an Influence Coefficient Method. *Appl. Ocean Res.* **2022**, *125*, 103235. [CrossRef]
18. Xu, Y.; Wu, J.; Li, P.; Kujala, P.; Hu, Z.; Chen, G. Investigation of the Fracture Mechanism of Level Ice with Extended Finite Element Method. *Ocean Eng.* **2022**, *260*, 112048. [CrossRef]
19. Zheng, X.; Tian, Z.; Xie, Z.; Zhang, N. Numerical Study of the Ice Breaking Resistance of the Icebreaker in the Yellow River Through Smoothed-Particle Hydrodynamics. *J. Mar. Sci. Appl.* **2022**, *21*, 1–14. [CrossRef]
20. Schwarz, J.; Frederking, R.; Gavrillo, V.; Petrov, I.G.; Hirayama, K.I.; Mellor, M.; Tryde, P.; Vaudrey, K.D. Standardized Testing Methods for Measuring Mechanical Properties of Ice. *Cold Reg. Sci. Technol.* **1981**, *4*, 245–253. [CrossRef]
21. Timco, G.W.; Weeks, W.F. A Review of the Engineering Properties of Sea Ice. *Cold Reg. Sci. Technol.* **2010**, *60*, 107–129. [CrossRef]
22. Wang, G.; Zhang, D.; Xu, N.; Wang, S.; Yu, D.; Yue, Q. The Research of Extreme Ice Load for Flexible Vertical Structures in Sub-Arctic Environment. *Ekoloji* **2018**, *27*, 2033.
23. Rødtang, E.; Alfredsen, K.; Høyland, K.; Lia, L. Review of River Ice Force Calculation Methods. *Cold Reg. Sci. Technol.* **2023**, *209*, 103809. [CrossRef]
24. Neill, C.R. Dynamic ice forces on piers and piles. An assessment of design guidelines in the light of recent research. *Can. J. Civil. Eng.* **1976**, *3*, 305–341. [CrossRef]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Experimental and Numerical Prediction of Slamming Impact Loads Considering Fluid–Structure Interactions

Tao Lu ¹, Jiaxia Wang ¹, Kun Liu ^{1,*} and Xiaochao Zhao ²

¹ School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang 212003, China; 211210101111@stu.just.edu.cn (T.L.); jxwang66@yeah.net (J.W.)

² Marine Design and Research Institute of China, Shanghai 200011, China; zxc1719738018@163.com

* Correspondence: kunliu@just.edu.cn; Tel.: +86-135-1169-2085; Fax: +86-0511-8444-6543

Abstract: Slamming impacts on water are common occurrences, and the whipping induced by slamming can significantly increase the structural load. This paper carries out an experimental study of the water entry of rigid wedges with various deadrise angles. The drop height and deadrise angle are parametrically varied to investigate the effect of the entry velocity and wedge shape on the impact dynamics. A two-way coupled approach combining CFD method software STAR-CCM+12.02.011-R8 and the FEM method software Abaqus 6.14 is presented to analyze the effect of structural flexibility on the slamming phenomenon for a wedge and a ship model. The numerical method is validated through the comparison between the numerical simulation and experimental data. The slamming pressure, free surface elevation, and dynamic structural response, including stress and strain, in particular, are presented and discussed. The results show that the smaller the inclined angle at the bottom of the wedge-shaped body, the faster the entry speed into the water, resulting in greater impact pressure and greater structural deformation. Meanwhile, studies have shown that the bottom of the bow is an area of concern for wave impact problems, providing a basis for the assessment of ship safety design.

Keywords: drop tests; slamming pressure; fluid–structure coupling; numerical simulation; hydroelasticity

1. Introduction

When ships navigate through the sea, they often encounter adverse weather conditions that require sailing through rough waves. The phenomenon of slamming that occurs when the hull collides fiercely with the waves can seriously damage the structural integrity of the ship and pose a threat to the life safety of maritime personnel. In 1994 [1], the Estonia sank because the wave loads it encountered exceeded the structural limits the vessel could withstand, leading to the damage of the positioning devices and the lock structures of the foredeck shelter-type bow doors. When encountering violent waves, the bow of the ship emerges from the sea and then re-enters the waves at a relatively high speed, colliding with the waves to produce the so-called slamming phenomenon. Based on the different areas affected by the slamming load, slamming can be classified into three types: bow flare slamming; stern slamming; and bottom slamming, with each type of slamming causing different dynamic responses [2].

For the exploration of slamming issues, scholars have proposed various theoretical and experimental methods for in-depth study. Ochi [3] proposed a correlation between the length of the hull and the slamming duration, adopting a quasi-static approach to study the dynamic response of the hull beam under slamming loads. Yu Pengyao et al. [4] used three-dimensional linear potential flow theory and long-term analysis methods of slamming speed to explore the direct calculation method of design loads for bow flare slamming pressure on ships. Hong et al. [5] conducted research on the dynamic characteristics of bow flare slamming in regular and irregular wave conditions, discussing the spatial distribution

and temporal progress of slamming loads on the bow flare. The experimental studies of hull models in wave tanks are of significant guidance and can provide a better understanding of the phenomenon of wave slamming. Hermundstad et al. [6] proposed a theoretical method that can effectively predict the slamming loads acting on the hull beam and verified the accuracy of this prediction method through model tests. Wang et al. [7] conducted experimental studies to investigate the bow flare slamming and bottom slamming phenomena of chemical tankers under the action of irregular waves and measured the probability and values of slamming loads, which has important practical value for ship design. Lindemann et al. [8] conducted research on the dynamic failure characteristics of ship structures under longitudinal and transverse slamming loads through experimental methods, considering the effects of inertia and damping, and improved the existing ideal structural unit method. Wang Xueliang et al. [9,10] took a large LNG ship as the research object and compared the wave-induced vibration response of the ship using tank model experimental methods and three-dimensional linear hydroelastic theory, which indicated that the wave slamming loads often cause issues of ultimate strength and fatigue damage in the ship structure. Jiao Jialong et al. [11] proposed a large-scale segmented-model wave-load testing technique in real sea wave environments and conducted hydroelastic tests on large-scale segmented ship models.

In the area of numerical simulation, Zhu Renqing et al. [12] conducted numerical simulations on the slamming problem of wedge-shaped bodies entering the water and obtained the effects of wedge stiffness on the water entry process. Yang et al. [13] studied the dynamic ultimate strength of hull beams under slamming bending moments through numerical simulation methods. The results showed that the duration, amplitude, and impulse of the slamming affect the dynamic response of the hull beam. Mackie [14] considered the influence of the fluid-free surface in the study of the two-dimensional rigid wedge entering the water and transformed it into a similar flow problem on the complex plane, thereby deriving the shape of the free surface and the slamming pressure. Bilandi et al. [15,16] used a combination of the finite volume method and the volume of fluid method to numerically simulate the vertical water entry impact of two-dimensional symmetrical and asymmetrical wedge structures. They also analyzed the hydrodynamic behavior of stepped planing hulls in eight different configurations using CFD methods. The results indicated that specific step heights and placements can significantly reduce the resistance of the vessel, thereby enhancing the performance of high-speed planing boats. Vesselin et al. [17] used Open FOAM to predict the asymmetric water entry slamming pressure of two-dimensional wedges and the history of liquid surface changes. Zhao Zhongbang [18] predicted the slamming loads for a typical position at the bow of a VLCC ship using a two-step method. The study established a three-dimensional uniform cross-sectional model and calculated slamming pressure curves and peak values at each point. Zhang Boran [19] simulated the motion of a three-dimensional hull through numerical waves, compared the pressure data of various points during the slamming process under head-on and oblique waves laterally, and obtained the trend of bow pressure distribution under different navigation conditions. Stavovy [20], based on the assumption that the slamming speed is equal to the relative speed between the moving body and the wave-slamming surface, derived a method for calculating slamming pressure applicable to all types of hulls. Peng Dandan [21] studied the impact of different parameters on the dynamic response of two-dimensional and three-dimensional structures, discussing the effects of structural plate thickness and changes in the elastic modulus on slamming pressure. Wang Jiaxia et al. [22] used STAR CCM+ to study the three-dimensional nonlinear wave slamming loads and conducted numerical predictions and hull optimization for the slamming loads in the bow and stern regions of a large cruise ship navigating in waves. Chillemi et al. [23] used CFD methodology simulations and parametric optimization algorithms to improve the form design of a racing motorbike. It was shown that the optimized design significantly reduces air resistance and improves downforce, thus enhancing the track performance of the motorbike. Lee et al. [24] analyzed the structural response of two types of planing hull

grillage panels in irregular waves through experiments and numerical simulations. The results showed that the greatest impacts occur in deep troughs with low wave heights. Numerical simulations more accurately predict the vertical acceleration of the center of gravity compared to traditional methods, demonstrating the efficiency of computational approaches and the conservatism of current design practices.

Overall, this thesis employs experimental and numerical simulation methods for its study. The experimental part mainly uses water entry experiments of wedges with different inclination angles to simulate the slamming process of ships in waves, examining how changes in entry speed and wedge inclination affect the slamming impact. The numerical simulation involves the coupling of CFD and FEM, utilizing STAR-CCM+12.02.011-R8 and Abaqus 6.14 software for coordinated two-way fluid–structure interaction analysis, allowing multiple data exchanges per time step until convergence criteria are met, thereby enhancing the calculation’s stability and accuracy. This coupling method can more accurately predict the pressure distribution during the slamming process, the elevation of the free surface, and the dynamic structural response. It provides a scientific basis for ship design and has an important reference value for assessing ship safety under extreme sea conditions. This research not only enhances our understanding of ship slamming response but also advances related numerical simulation technologies.

2. The Numerical Methods

2.1. Governing Equations

The finite volume method (FVM) was imported into STAR-CCM+ software, and the integral form of the governing equation was discretized into a system of algebraic equations in the time and space dimensions. First, the calculation domain was divided into a limited number of adjacent control bodies. These control bodies can be of any polyhedron shape. The discrete governing equations also need to use area integration, volume integration, and time and space derivatives in the calculation process.

It was assumed that the flow is controlled by the RANS equation for viscous three-dimensional flow, where the turbulence effect includes a vortex model and a viscous model. At this time, the continuity equation, momentum equation, and two turbulence characteristics equations needed to be solved. The model selected the Realizable K-Epsilon turbulence model to simulate the effect of turbulence in the fluid, which added some improvements to better handle turbulent flows compared to the Standard K-Epsilon model for high Reynolds number flows, where the governing equations of mass conservation and momentum conservation in integral form can be written as follows:

Conservation of mass:

$$\frac{d}{dt} \int_V \rho dV + \int_S \rho (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS = 0 \quad (1)$$

Conservation of momentum:

$$\frac{d}{dt} \int_V \rho \mathbf{v} dV + \int_S \rho \mathbf{v} (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS = \int_S (\mathbf{T} - p\mathbf{I}) \cdot \mathbf{n} dS + \int_V \rho \mathbf{b} dV \quad (2)$$

where ρ is the density of water, $\mathbf{v}$ is the fluid velocity vector, $\mathbf{v}_b$ is the surface velocity of the control body; $\mathbf{n}$ is the unit vector normals perpendicular to the surface of the control body, where S and V are the area and volume of the control body, respectively, $\mathbf{T}$ is the stress tensor (representing the velocity gradient and eddy viscosity), p is the pressure, $\mathbf{I}$ is the unit tensor.

The volume of fluid model assumes that all immiscible fluids in the control body have the same velocity, pressure, and temperature fields, so only the basic governing equations for the conservation of momentum, mass, and energy of a single-phase flow field in the

control body require solving. The conservation equation describing the volume fraction α_i is

$$\frac{d}{dt} \int_V \alpha_i dV + \int_S \alpha_i (v - v_g) da = \int_V \left(s_{\alpha_i} - \frac{\alpha_i}{\rho_i} \frac{D\rho_i}{Dt} \right) dV \quad (3)$$

where s_{α_i} is the source or sink of phase i flow field, $D\rho_i/Dt$ is the material derivative of phase density ρ_i .

2.2. Structural Response Equation

The dynamic response of a three-dimensional ship under a slamming load was investigated through the finite element method, and the effect of the elastic structure response on the slamming load was also taken into account. The three-dimensional model was discretized to obtain a finite element model of the structure. This is also a basic step in finite element calculation. The basic equation of the three-dimensional elastic dynamic equation is based on obtaining the finite element model of the hull structure, and the finite element solution steps for the three-dimensional solid dynamic analysis, which are as follows:

$$[M] \{\ddot{u}\} + [C] \{\dot{u}\} + [K] \{u\} = [P] \quad (4)$$

where $[M]$ is the mass matrix; $[C]$ is the damping matrix; $[K]$ is the stiffness matrix; $[P]$ is the external incentive; $\{\ddot{u}\}$ is the node acceleration vector; $\{\dot{u}\}$ is the node speed vector; $\{u\}$ is the node displacement vector.

2.3. Fluid–Structure Interactions

The wave-slamming problem of the hull structure belongs to the strong nonlinear, fluid–structure interaction problem. In this paper, the data were automatically transferred at the fluid–solid interface through the cooperative coupling function between software STAR-CCM+ and Abaqus to realize the two-way fluid–structure interaction calculation. STAR-CCM+ solves the fluid equation, and Abaqus solves the structural equation, which belongs to the partitioned fluid–structure interactions. In this article, coupling calculations were coupled using an implicit method between STAR-CCM+ and Abaqus, mainly because the implicit coupling method is suitable for strong coupling calculations between structures and fluids. It allows STAR-CCM+ and Abaqus to exchange data more than once each time and keep iterating until the convergence criteria are met. This coupling scheme is more stable, and the second-order accuracy is higher.

Since the wave-slamming phenomenon is a strong coupling phenomenon, an implicit strong coupling algorithm was selected in the numerical model. As shown in Figure 1, a single coupling procedure was taken as an example to introduce the process of the implicit coupling time step. Set the motion specification as ‘deformation’ in STAR-CCM+, and set the elastic modulus, plate thickness, initial speed, gravity, time step, and coupling surface of the model in Abaqus. The specific calculation process was as follows: STAR-CCM+ starts the flow field load calculation, then transmits the pressure and shear force to the FEM solver, and then Abaqus performs structural response analysis based on the load of the coupling interface and transfers the displacement and deformation to STAR-CCM+. STAR-CCM+ then moves the mesh by the ‘deformation’ function according to the amount of displacement transmitted. This completes a coordinated coupling for calculation and update and repeats until the maximum physical time is reached.

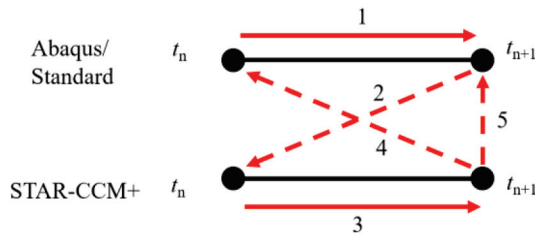


Figure 1. The procedure for a single coupling step with implicit solvers.

3. Model Tests

3.1. Test Models and Test Point Arrangement

The research object of this test was a wedge. The wedge model was 600 mm long and 300 mm wide with a fixed aspect ratio. The shape of the bottom section of the model was V-shaped (wedge). The inclination angles of the bottom plate (β) were 10° , 20° , 30° , and 40° . The detailed dimensions of the two-dimensional wedge model are shown in Figure 2a.

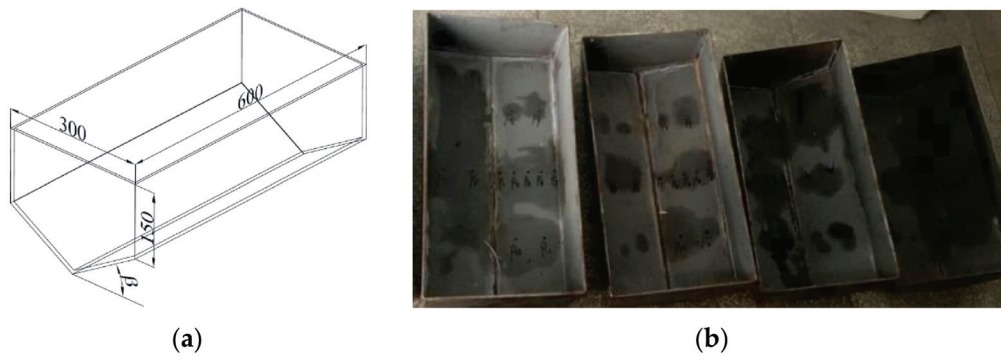


Figure 2. Size of test models (Unit: mm). (a) model shape. (b) model with different deadrise angles.

In order to make the fluid flow two-dimensionally during the impact of the wedge into the water, the structure of the model remained unchanged in the length direction. In addition, as shown in Figure 2b, in order to give the structure a large buoyancy reserve after entering the water, the wedge-shaped structure was designed with vertical baffles added around it.

In order to measure the slamming pressure at different positions on the base plate of the test model, the measuring points were arranged at different height positions on the bottom plate, and the measuring points were compared to corresponding measuring points. The specific distribution of the wedge measuring points is shown in Figure 3. The center of the bottom plate (the angle of inclination with respect to the horizontal plane) was O point, and the measuring point P1 was arranged at a distance of 15 mm from the O center on the lateral centerline of the bottom plate and then one measuring point was arranged every 30 mm; that is, P2, P3, P4, P5, P6, and P7 in Figure 3a are the comparative measuring points of P2 and P4 in the horizontal direction (the other side of the bottom plate). P8, P10, P9, and P11 are the measurement points of P2 and P4 in the longitudinal direction for comparison. In addition, the openings in the bottom hole of test models and the fitting arrangement of pressure sensors are shown in Figure 3b.

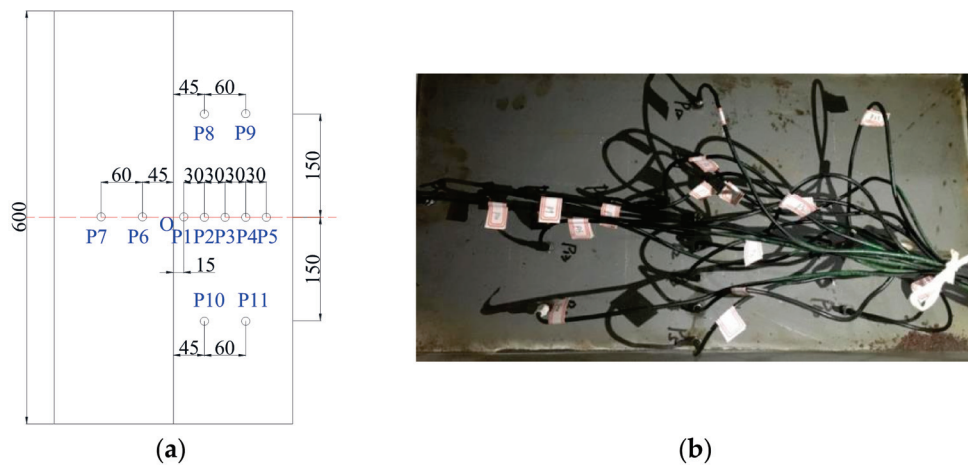


Figure 3. Distribution and numbering of measure points on wedge bottom (Unit: mm). (a) Measure points on the wedge bottom. (b) Real scenery of pressure sensors.

3.2. Test Equipment and Devices

This test was carried out in a towing pool of the Jiangsu University of Science and Technology. The test piece model was suspended on the crane hook by a non-retractable steel cable. The initial drop height was determined by the crane lifting and the laser rangefinder (the determination of the test height was based on the distance from the lowest point of the test piece to the water surface). After adjusting to the specified height, the hook was released, and the test model was freely released into the water. In the test process, a high-speed camera photographed the test model's release into the water. The signal acquisition system collected and recorded the test data measured by the pressure sensor through a wire. The test scenario for water entry of the structure is shown in Figure 4.

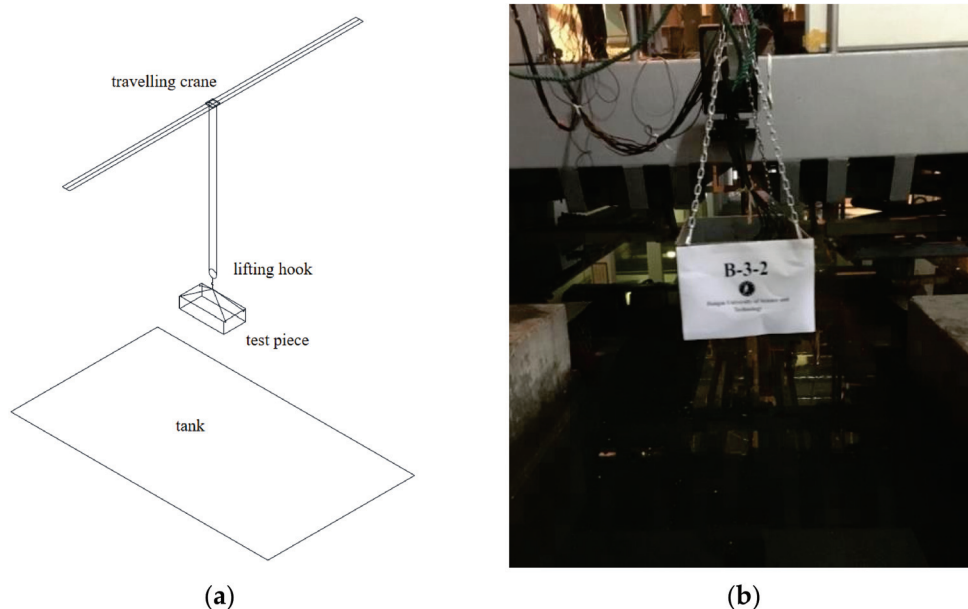


Figure 4. Test scenario for water entry of structure. (a) Schematic diagram of the test scenario. (b) Actual test scenario.

The test acquisition equipment used during the test was as follows: (1) Laser rangefinder: measuring distance 750 mm, the sampling period was selected from 20/50/100/200/500/1000 μ s, measuring accuracy was 2 μ m; (2) High-speed camera: Model Phantom V12.1, sensor resolution was 1280 $\times$ 800, equipped with EDR dynamic range exposure control function and

automatic exposure function, full-frame 1280×800 , speed 6242 frames/second; (3) Pressure sensor: HM91 micro sensor and transmitter, the diameter was 6 mm, the sampling frequency was 10,000 Hz; (4) Dynamic signal acquisition system: two ruifeng real-time collectors, 48 channels, the sampling period was 0.5 microseconds.

During the processing of sample models, the thickness of the model steel plate was 5 mm. The quality and the corresponding number of each structural model after processing are given in Table 1. The initial height of water entry according to test conditions was determined to obtain the test cases. Each test condition was repeated three times.

Table 1. Test cases for water entry of models.

Model	Mass/kg	Case 1	Case 2	Case 3	Case 4	Case 5	Case 6	Case 7
		0.1 m	0.2 m	0.3 m	0.4 m	0.6 m	0.8 m	1.2 m
Wedge $\beta = 10^\circ$	18.5	✓	✓	✓	✓	---	---	---
Wedge $\beta = 20^\circ$	19.2	✓	✓	---	✓	✓	---	---
Wedge $\beta = 30^\circ$	20.2	---	✓	---	✓	✓	✓	---
Wedge $\beta = 40^\circ$	22.1	---	✓	---	✓	✓	✓	---

3.3. Analysis of Test Results

According to the above test cases, the slamming test of the wedge-shaped structure was carried out to obtain the curve of the slamming pressure with time at each measurement point. In order to facilitate comparison and analysis, the time at which the test model was at a height of 0.01 m from the water surface was taken as the time 0.

3.3.1. Slamming Pressure at Different Deadrise Angles

Curves of slamming pressure at different measurement points on wedges with deadrise angles β of 10° , 20° , 30° , and 40° over time are shown in Figure 5. It can be seen from the figure that the wedge with a deadrise angle of 10° (Figure 5a) reached the peak value of the slamming pressure immediately when it contacted the water surface. Over time, each measurement point had a significant peak value in succession and then rapidly decreased to a stable value, and the slamming pressure peaks at the measurement points P1 to P5 showed a trend of increasing first and then decreasing. In Figure 5b, the peak pressure of the slamming pressure at the measuring points P1 to P5 of the wedge with a deadrise angle of 20° also increased first and then decreased, but the position where the peak value of the maximum slamming pressure moved to the P2 measurement point. The peak slamming pressure of the wedges with deadrise angles of 30° (Figure 5c) and 40° (Figure 5d) gradually decreased from the measurement points P1 to P5. Unlike the first two types of wedges, the peak value of the maximum slamming pressure appeared at the P1 measurement point. This is mainly because when the wedge body enters the water, the bottom of the wedge with a small deadrise angle will cause a greater obstacle to liquid surface lifting, so the liquid surface continues to rise, and the squeezing force of the bottom plate on the liquid surface continues to increase, and the slamming pressure also increases continuously. After the liquid surface is lifted to a certain height, a jet phenomenon will occur, and the slamming pressure peak will appear at the root of the jet. After that, the jet is separated from the surface, and the slamming pressure decreases rapidly.

Comparing the slamming pressure curves of Figure 5a–d, it can be found that the slamming pressure of the wedge changed with the change in the deadrise angle: the deadrise angle increased, and the slamming pressure decreased significantly. This is because the normal direction of the wedge force (the direction perpendicular to the surface of the wedge body) had a larger component when the wedge had a small deadrise angle, so the slamming pressure on the wedge was also significantly larger. In addition, the slamming pressure of wedges with different deadrise angles had different degrees of oscillation. When the deadrise angle was small, the oscillation frequency was low and the amplitude was large; when the deadrise angle was large, the oscillation frequency

was high and the amplitude was small. This is because the difference in the deadrise angle resulted in changes in the amplitude and frequency of the shock wave. Table 2 shows the peak slamming pressure of the comparative measurement points of wedges with different deadrise angles and their occurrence time (due to the damage of the P7 measuring point during the test, individual data are not given). It can be found that the peak value of slamming pressure at the P2 and P4 measurement points and their comparative measurement points were almost the same by comparison. However, when the deadrise angle was 10° and 30° , the peaks value of the P6 and P7 measurement points appeared earlier than P2 and P4, and the peak value of the slamming pressure was too large. When the deadrise angle was 30° , the peaks value of P8 and P9 measurement points appeared later than P2 and P4, and the peak value of the slamming pressure was smaller. This is mainly because the wedges tilted at a small angle during the test, which resulted in the measuring points at the same height not entering the water at the same time, thus affecting the slamming pressure. However, since the wedge was tilted at an extremely small angle, the effect of test errors can be ignored.

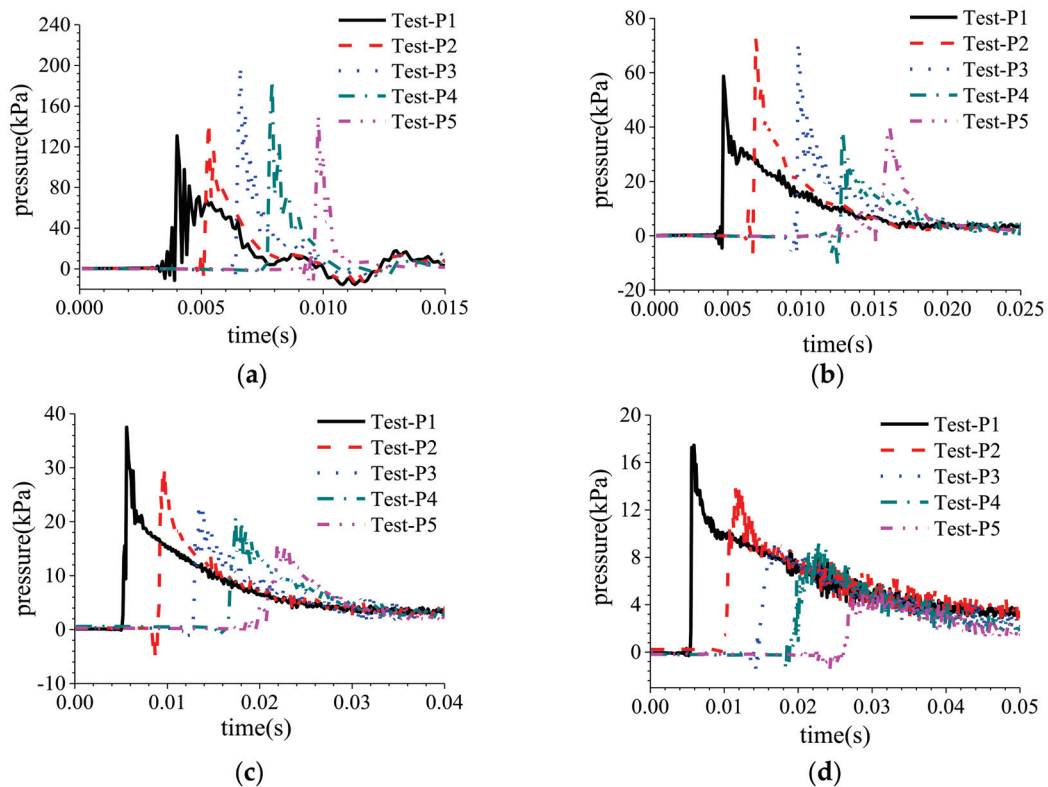


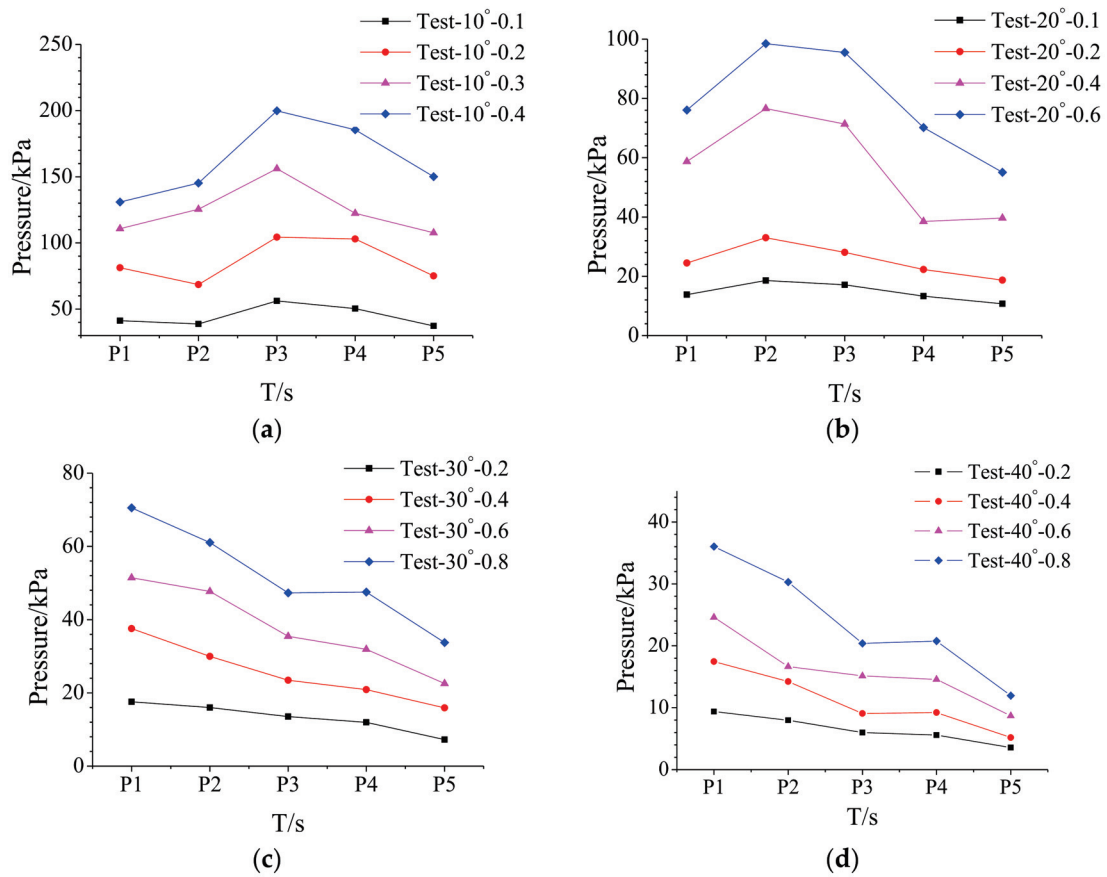
Figure 5. Pressure of wedges with different deadrise angles β dropped from 0.4 m height. (a) $\beta = 10^\circ$. (b) $\beta = 20^\circ$. (c) $\beta = 30^\circ$. (d) $\beta = 40^\circ$.

3.3.2. Slamming Pressure at Different Entry Speeds

Figure 6 shows the peak slamming pressure of the measurement points of the wedges at different deadrise angles at different drop heights. It can be seen from the figure that as the drop height increased, that is, the speed of entering the water increased, and the peak value of the slamming pressure also increased. However, the peak value of slamming pressure decreased as the deadrise angle increased for wedges with different deadrise angles falling from the same height.

Table 2. Comparison of peak slamming pressure on contrastive measure points.

Deadrise Angle β		P2	P6	P8	P10
10°	T/s	0.0053	0.0052	0.0058	0.0052
	P/kPa	145.296	161.111	148.249	151.882
20°	T/s	0.0069	0.007	0.0073	0.0067
	P/kPa	72.581	78.82	77.36	75.37
30°	T/s	0.0097	0.0089	0.0108	0.0103
	P/kPa	29.983	32.905	28.895	30.15
40°	T/s	0.0114	0.0098	0.0106	0.0105
	P/kPa	14.223	13.947	12.27	12.976
Deadrise Angle β		P4	P7	P9	P11
10°	T/s	0.0079	0.0074	0.0087	0.0078
	P/kPa	185.57	189.983	168.033	171.513
20°	T/s	0.0128	---	0.0132	0.013
	P/kPa	38.541	---	42.676	43.546
30°	T/s	0.0174	0.0158	0.0189	0.0185
	P/kPa	20.909	23.937	17.465	18.714
40°	T/s	0.0244	---	0.0247	0.0238
	P/kPa	9.2251	---	9.1463	8.184

**Figure 6.** Peak pressure of the wedge with different water entry heights. (a) $\beta = 10^\circ$. (b) $\beta = 20^\circ$. (c) $\beta = 30^\circ$. (d) $\beta = 40^\circ$.

Comparing the measured points of the tip when the wedges with different deadrise angles had the maximum slamming pressure, it can be found that when the deadrise angle was 10° , the maximum slamming pressure tip appeared at the P3 measurement point and

when the deadrise angle was 20° , the maximum slamming pressure appeared at the P2 measurement point. The maximum slamming pressure appeared at the P1 measurement point when the deadrise angle was 30° and 40° . The phenomenon is because when the deadrise angle was small, the liquid surface lift and the jet phenomenon had a greater effect on the peak value of the slamming pressure. When the deadrise angle was large, the peak value of the slamming pressure was hardly affected.

4. Simulation Analysis of a Wedge-Shaped Structure Slamming into Water

4.1. Verification of Numerical Methods

This section is mainly divided into two parts. The first is the comparison between the numerical model and the experiment result of water entry of a wedge. In this numerical model, the deformation effect of the structure was taken into account through the co-simulation of STAR-CCM+ and Abaqus. The other part is the comparison with the experimental results of the ship model in regular waves in reference [25]. In this part, in order to be more consistent with the experimental scene, the rigid body motion of the ship model was considered through the DFBI model.

4.1.1. Computational Model

The main scale parameters of the experimental model are shown in Table 3. The size of the overall calculated background area was $0.8\text{ m} \times 0.8\text{ m} \times 0.8\text{ m}$, and that of the overlapping area was $0.4\text{ m} \times 0.7\text{ m} \times 0.35\text{ m}$. The origin point of the coordinate system was at the midpoint of the wedge bottom, the Z axis was positive vertically, and the X axis was positive horizontally to the right, which satisfied the right-hand rectangular coordinate system, in which the height of the water surface was at $Z = -0.01\text{ m}$. For the setting of boundary conditions, the background area adopted the wall boundary condition, and the boundary condition of the top was the pressure outlet, the overlapping area of the wedge adopted the wall, and the rest adopted the overlapping mesh.

Table 3. Principal dimension.

Principal Scale	Parameter Value
Length overall (m)	3.8
Length at waterline (m)	3.32
Length of parallel body (m)	3.0
Beam (m)	0.75
Draught (m)	0.08
Depth (m)	0.2
Volume of displacement (m^3)	0.13
Area of water-plane (m^2)	2.38

The origin of the coordinate system was at the intersection of the vertical line from the stern and the plane of the ship's bottom. The X-axis pointed toward the bow along the length of the ship, and the Z-axis was perpendicular and pointed upward, satisfying the right-hand Cartesian coordinate system. The coordinates for the background domain area were $[-8, -4, -4]$ and $[24, 4, 4]$, while the overlapping region coordinates were $[-1.5, -1.2, -0.5]$ and $[5, 1.2, 0.8]$, as shown in Figure 7.

The boundary condition settings for the background domain area and the overlapping region can be referred to in Table 4.

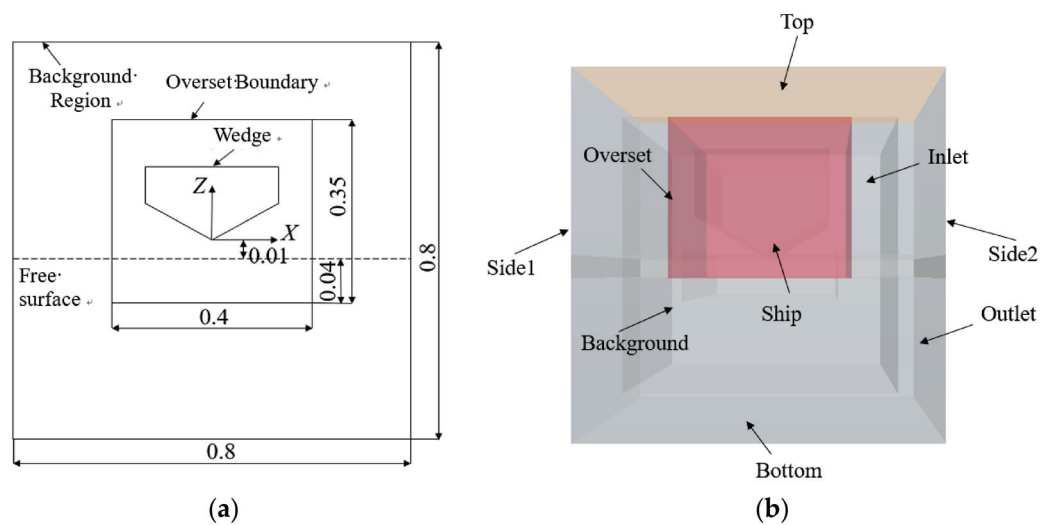


Figure 7. Computational domain profile for simulation. (a) Computational domain for wedge. (b) Computational domain for ship.

Table 4. Boundary conditions of background and overset region in computational domain.

Region	Boundary	Boundary Condition
Background	Top	Wall
	Bottom	Wall
	Side1	Symmetry
	Side2	Symmetry
	Inlet	Velocity Inlet
	Outlet	Pressure Outlet
Overset	Overset Part	Overset Mesh
	Hull	Wall

4.1.2. Grid Division and Time Step Setting

Due to the structure of the wedge involved the coupling of three-phase substances in the process of slamming, it was necessary to further track the large displacement movement of the structure and the free liquid surface, and the deformation of the structure made it necessary to reconstruct the mesh at each time step, which increased the complexity of calculation. To solve this problem, this paper introduced the three-dimensional overset mesh technology, which can consider the large deformation of the structure and is suitable for solving this kind of complex fluid–structure coupling problem without mesh reconstruction. In the computing area, the methods of surface reconstruction, the repair of the automatic surface, cutting mesh, and prismatic layer were used to generate meshes. In order to eliminate the errors caused by inserting variables between meshes, the same mesh density of the magnitude was used for the overlapping area of the background mesh and the overlapping mesh. See Figure 8a for the mesh division of the calculation area. The maximum mesh size was 0.02 m, the encrypted mesh size was 0.008 m, and the cutting body mesh unit was isotropic. There will be jet flow in the solid-liquid interface. In order to clearly capture the evolution form of jet flow, this chapter used mesh thinning technology to obtain fine mesh at the interface. The mesh density between the contact water surface and the wedge bottom is shown in Figure 8. The cutting body mesh unit was anisotropic. The mesh size near the water surface was 0.008 m in X and Y directions and the size in the Z direction was 0.008 m. The mesh size in X and Y directions in the falling area of the wedge bottom was 0.004 m. The mesh size of the overlapping mesh area was 0.004 m, and that of the V-shaped plate at the bottom of the wedge was 0.004 m.

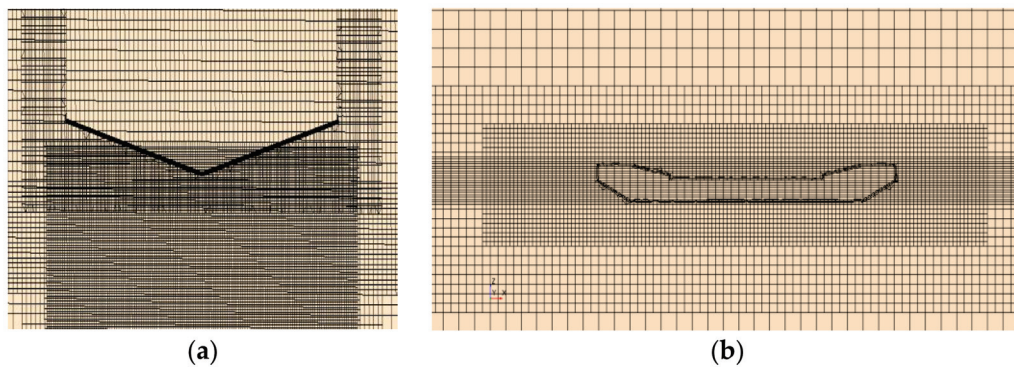


Figure 8. Computational domain mesh. (a) Mesh density between water and wedge. (b) Mesh density between water and ship model.

In the process of numerical calculation, the Courant number c was used to balance the accuracy of calculation results and calculation efficiency. The Courant number indicates how many meshes have moved in the unit time step. After correct adjustment, the convergence can be accelerated, and the stability of the solution can be enhanced. The Courant number was 1 when first-order convection was used, and the Courant number was 0.5 when second-order convection was used. In this section, the time step was 2.0×10^{-6} s.

$$C = \frac{U\Delta t}{\Delta x} \quad (5)$$

where Δt is the time step length, Δx is the mesh size, and U is the wave velocity.

4.2. Calculation Results and Analysis

4.2.1. Comparison of Slamming Pressures for Wedges with Different Angles

The slamming values for wedges with different angles can be seen in Figure 9, which shows the slamming-pressure curve of a wedge falling from a height of 0.4 m. As can be seen from the graph, at an angle of 10° , the wedge (Figure 9a) had the highest slamming pressure when the slamming first occurred, with a large slamming pressure at the first measurement point on the wedge. Figure 9b shows a similar trend to that of the 10° wedge in terms of peak slamming pressure at measurement point P4 with an angle of 20° , which also increased and then decreased. This is due to the fact that when the wedge dropped into the water, the bottom plate of wedges (10° and 20°) affected the water, so when the liquid level was continuously lifted, the free liquid surface was subjected to increasing squeezing pressure from the bottom plate, and therefore the slamming pressure also increased at this time. Once the free liquid surface rose to a certain height, wave breaking and peak of the slamming pressure appeared. This was followed by a gradual reduction in the slamming, which was caused by the separation of the wedge surface from the free liquid surface. And with the increase in the angle of the wedge, the rise in the free liquid surface was not affected too much by the base plate.

As can be seen in Figure 9, the slamming pressure of the wedge changed as the angle of the wedge changed, and as the angle increased, the slamming pressure of the wedge decreased. A comparison of the values of the experimental and simulated values is shown in Table 5. As can be seen from the table, the results of the experimental and simulated values were close, with a small error rate of around 20%. The error in the results for the wedge of 10° was larger because the wedge was subjected to a higher air resistance, which was not considered in the numerical simulation.

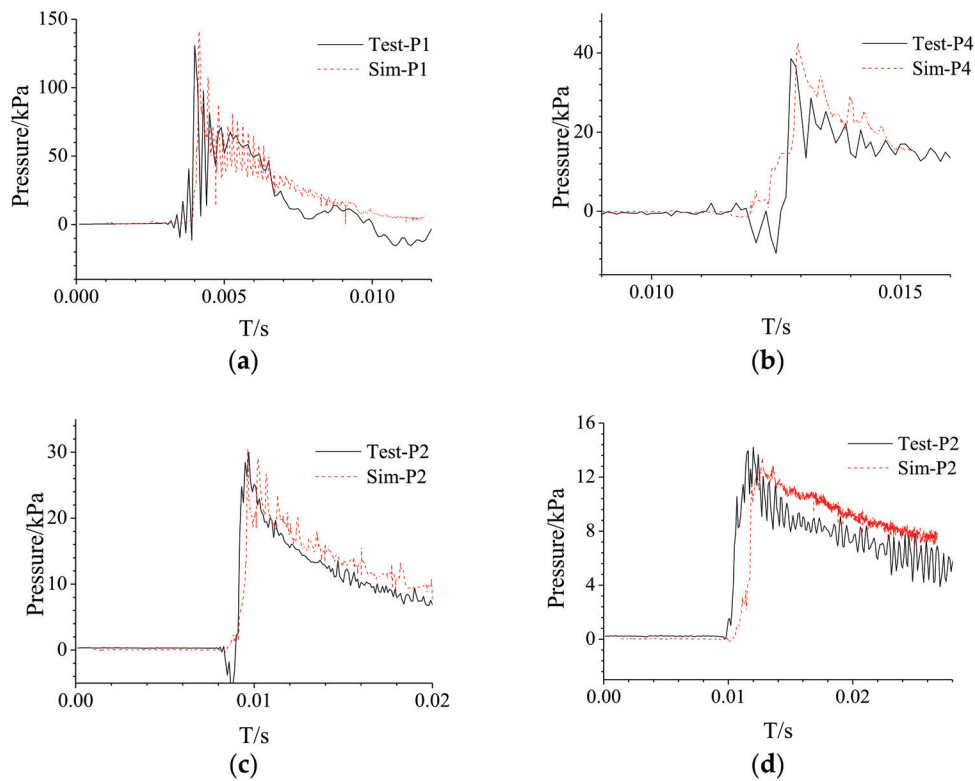


Figure 9. Comparison of the predicted time history of pressure against experimental results for water entry of wedge with constant deadrise angle β . (a) $\beta = 10^\circ$ (b) $\beta = 20^\circ$. (c) $\beta = 30^\circ$. (d) $\beta = 40^\circ$.

Table 5. Comparison of simulation and experiment.

β		P1 (P/kPa)	P2 (P/kPa)	P3 (P/kPa)	P4 (P/kPa)
10°	Experiment	130	144	200	184
	Simulation	141.261	/	149.988	143.465
20°	Expiment	56	72	68	38
	Simulation	61.001	57.8324	53.5308	43.2922
30°	Expiment	38	30	24	22
	Simulation	31.8174	30.4179	21.2797	18.6262
40°	Experiment	17.6	14	8.8	9
	Simulation	14.0357	13.3962	11.7048	9.04

4.2.2. Comparison of Slamming Results for Different Heights of the Wedge

Figure 10 shows the comparison of the slamming pressure of the wedges at P1, P2, P3, and P4 for different velocities (different heights). The larger angle of the wedge indicated a lower pressure.

When the angle of the wedge was 10° , the wedge at the P3 had the highest slamming pressure value. When the angle of the wedge was 20° , the wedge at the P2 had the highest slamming pressure value. When the angle of the wedge was 30° , the wedge at the P1 had the highest slamming pressure value. This is because as the angle became smaller, the slamming pressure was influenced more by the rise in free liquid, and the wave-breaking phenomenon became more pronounced, while for wedges with a larger angle, the rise in free liquid and the wave-breaking phenomenon had relatively little effect on the slamming pressure.

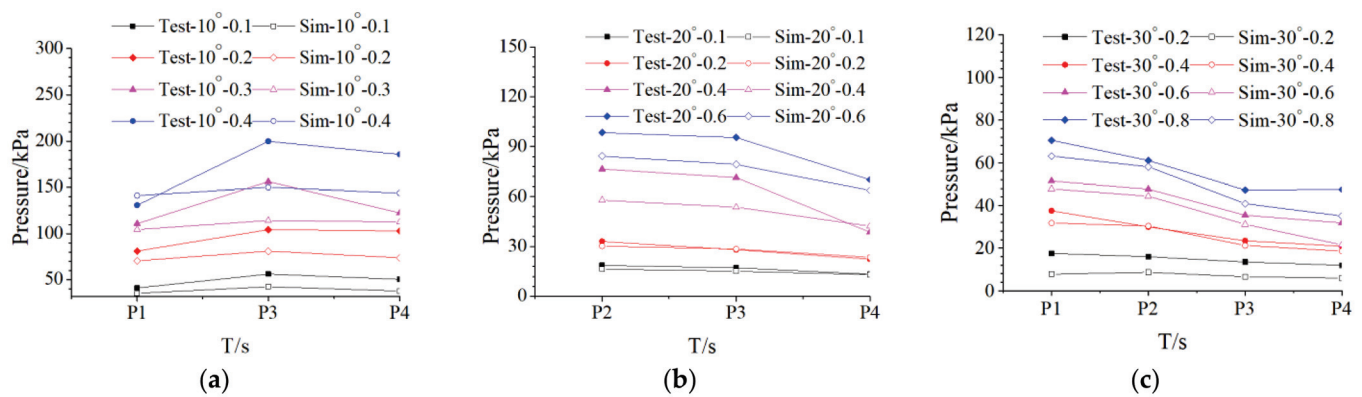


Figure 10. Peak slamming pressure under different angles. (a) $\beta = 10^\circ$ (b) $\beta = 20^\circ$ (c) $\beta = 30^\circ$.

In order to investigate the comparative slamming pressure of wedges at different water heights, comparative slamming pressure values are given in this section for wedges with a deadrise angle of 10° at 0.1, 0.2, 0.3, 0.4 m, for wedges with a deadrise angle of 20° at 0.1, 0.2, 0.4, 0.6 m, for wedges with a deadrise angle of 30° at 0.2, 0.4, 0.6, 0.8 m, and for wedges with a deadrise angle of 30° at 0.2, 0.4, 0.6, 0.8 m, respectively.

As can be seen from the graph, for the wedge of 10° , there was a certain error between the numerical simulation and the experimental peak slamming pressure, mainly because the wedge of 10° had a small angle and was susceptible to the influence of the jet when the liquid surface was lifted, so there was a certain error. For the wedges of 20° and 30° , the numerical simulations fit the experimental slamming pressure values better. Comparing the peak slamming pressures for wedges with different velocities at the same angle, it can be seen that the peak slamming pressure increased when the velocity became faster (the higher height).

4.2.3. Comparison of Slamming Results for the Ship Model

This section experiment investigated the slamming scenarios and structural response of an aluminum model ship under regular wave encountering conditions using a towing tank experiment. The experiments in the literature were conducted with 12 sets of calculations at velocities of 0, 1, and 1.5 m/s, wave frequencies of 0.56, 0.60, 0.64, 0.68, and 0.72 Hz, with wavelengths ranging from 3 m to 5 m. In this section, two sets of operating conditions (ship without speed and with speed) were selected for numerical simulation and compared with experimental results to validate the feasibility of the numerical method for wave-slamming response coupling in this section while analyzing the structural motion and slamming response characteristics. The selected two sets of operating conditions are shown in Table 6.

Table 6. Two working conditions selected for simulation.

Condition	Velocity (m/s)	Wave Height (m)	Wave Frequency (Hz)
1	0	0.23	0.56
2	1	0.15	0.56

The comparison between the vertical loads at the connection point of the support strut and the hull obtained from numerical simulations of the aluminum model and the measured vertical forces on the support strut during experiments is shown in Figure 11. From the figure, it can be observed that the overall trend of the simulation results aligned well with the experimental values, with a high degree of agreement. However, the magnitude of the simulated loads was slightly lower than the experimental values. This could be attributed to the fact that wave experiments were conducted in a finite water tank where wall reflections of waves might have led to an overestimation of the experimental results.

Additionally, the overall error for condition 1 was slightly higher than for condition 2, which could be due to the larger wave height in condition 1, resulting in a greater difference between the experimental and simulated values. Based on the load comparison curves in Figure 11, it can be concluded that the numerical simulation method proposed in this paper for the coupled response of the hull under wave slamming is effective.

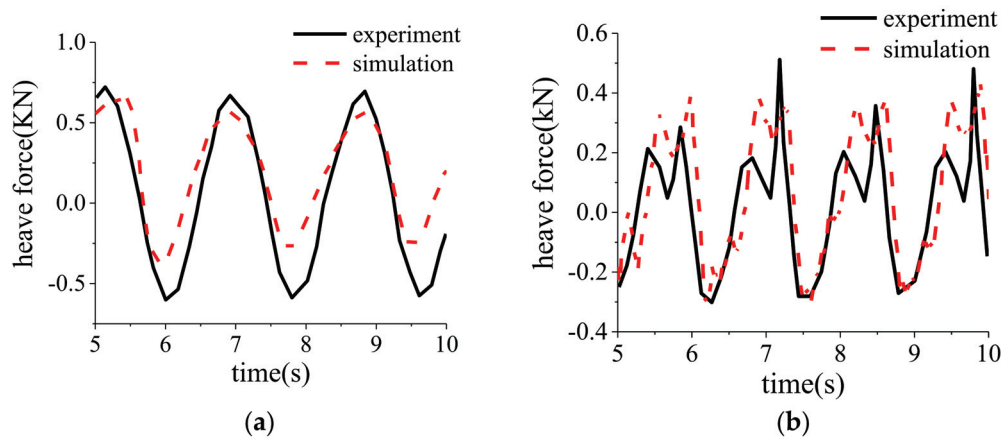


Figure 11. Comparison of experimental and numerical values of heave force. (a) Work condition 1. (b) Work condition 2.

4.3. Effect of Structural Response for the Wedge

To analyze the effect of the inclined angle of the wedge bottom on the dynamic response of the wedge entering the water, the inclined angle of the wedge was varied to 10° , 20° , 30° , and 40° , with the wedge impacting the water surface at a speed of 10 m/s. The slamming pressure curves at points P1, P3–P5, for different inclined angles are shown in Figure 12. From the figure, it can be observed that except for the curve at an inclined angle of 10° , the curves for other angles of the wedge were similar. They all show that at the moment of contact between the wedge and the water surface, the slamming pressure quickly reached its peak. With time, the slamming pressure at each measuring point rapidly decreased after reaching a significant peak. A smaller inclined angle at the bottom resulted in a larger slamming pressure, consistent with the trend of changes in the maximum slamming pressure peak for different inclined angles, as shown in Figure 13. However, at an inclined angle of 10° , the slamming pressure reached its peak at the moment of contact between the wedge and the water surface, then quickly decreased, followed by a rapid rise and fall, forming periodic variations. Periodic fluctuations were also present at an inclined angle of 20° but with smaller amplitudes. This is due to changes in structural characteristics caused by variations in the inclined angle of the wedge bottom. A smaller inclined angle at the bottom resulted in a more violent water impact upon entry and made it more difficult for air to escape, leading to periodic changes in slamming force. Conversely, when the inclined angle was larger, water entry obstruction was reduced, resulting in lower slamming pressure and smoother slamming curves.

Figure 13 depicts the variation curve of the maximum pressure peak of the wedge with increasing bottom inclination angles while keeping the plate thickness ($b = 5$ mm), elastic modulus ($E = 2.1 \times 10^{11}$ Pa), and water entry velocity ($v = 10$ m/s) constant. From the figure, it can be observed that as the bottom inclination angle increased, the maximum pressure value on the wedge gradually decreased. This is because increasing the bottom inclination angle reduced the obstruction to water entry for the structure, leading to a decrease in the slamming pressure experienced by the structure.

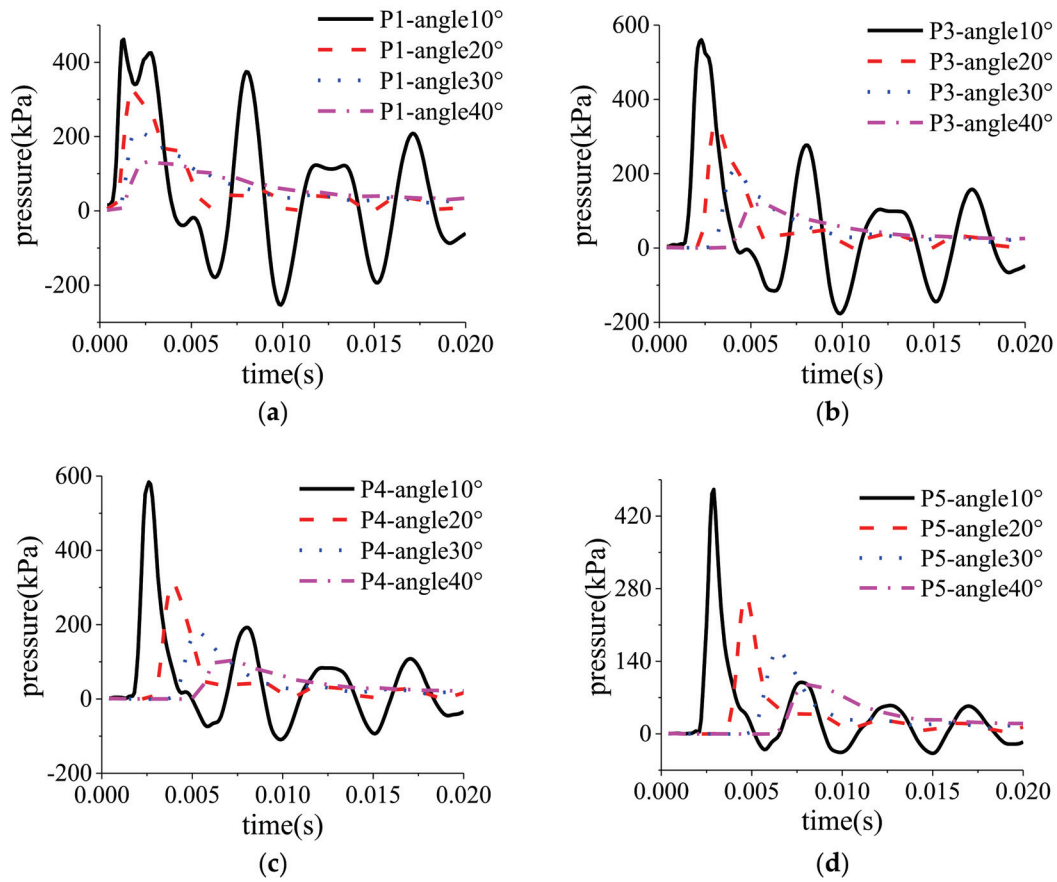


Figure 12. Comparison of slamming pressure of each point with different angles. (a) P1. (b) P3. (c) P4. (d) P5.

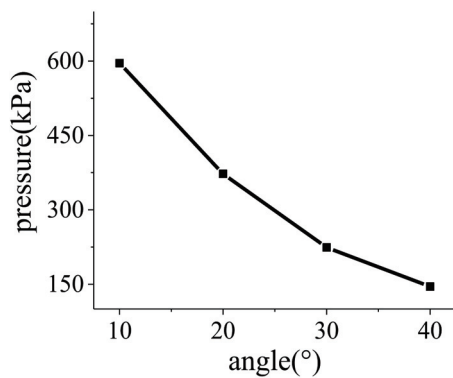


Figure 13. Comparison of maximum slamming pressure peak values of wedges under different angles.

Figure 14a is the deformation displacement curve of measure points P1 to P5 on the wedge with a bottom deadrise angle of 10°. It can be seen from the figure that when the deadrise angle was 10°, the deformation displacement value of each measure point had a large fluctuation. Among them, P5 and other measure points had opposite phase differences. The closer the measuring point was to the bottom tip, the larger the deformation displacement and fluctuation. The smaller the bottom deadrise angle of the wedge, the greater the slamming pressure, the greater the deformation displacement, and the greater the fluctuation of pressure value and deformation value. Figure 14b is a comparison of the deformation displacement of P1 point on the wedge at different bottom deadrise angles. It can be seen from the figure that the curve fluctuated greatly when the deadrise angle was

10°, the displacement curves of each measuring point at other deadrise angles increased with time, and the smaller the deadrise angle, the larger the peak value of displacement. This is because when the bottom deadrise angle of the wedge was larger, the structure had fewer obstacles to enter water, but when the deadrise angle was smaller, the wedge was more affected by air coupling; not only did the slamming pressure increase but also deformation fluctuations occurred easily.

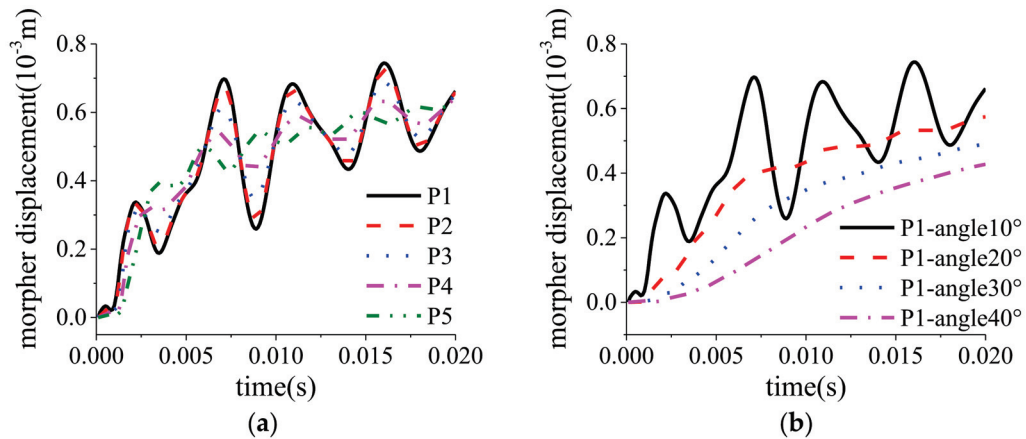


Figure 14. Deformation displacement of each measuring point at different angles. (a) $\beta = 10^\circ$. (b) P1.

When plate thickness ($b = 5$ mm), modulus of elasticity ($E = 2.1 \times 10^{11}$ Pa), and water velocity ($v = 10$ m/s) remained unchanged, the bottom deadrise angle increased. The change curve of the maximum deformation displacement of the wedge is shown in Figure 15. It can be seen from the figure that as the deadrise angle increased, the value of deformation displacement decreased continuously. This is because the larger the deadrise angle, the smaller the resistance to the structure, the smaller the deformation.

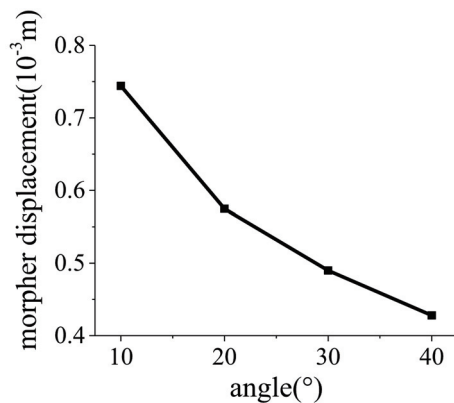


Figure 15. Comparison of maximum deformation displacement under different angles.

Figure 16 shows the stress nephogram of a wedge with a deadrise angle of 10° and a thickness of 5 mm at different times. It can be seen from Figure 16a that when the wedge just touched the water surface, the maximum stress was concentrated at the tip of the wedge. It can be seen from Figure 16b,c that during the process of entering water, the stress concentration starts to shift upward from the bottom tip, and finally appeared in the center of the plate at the top of the wedge.

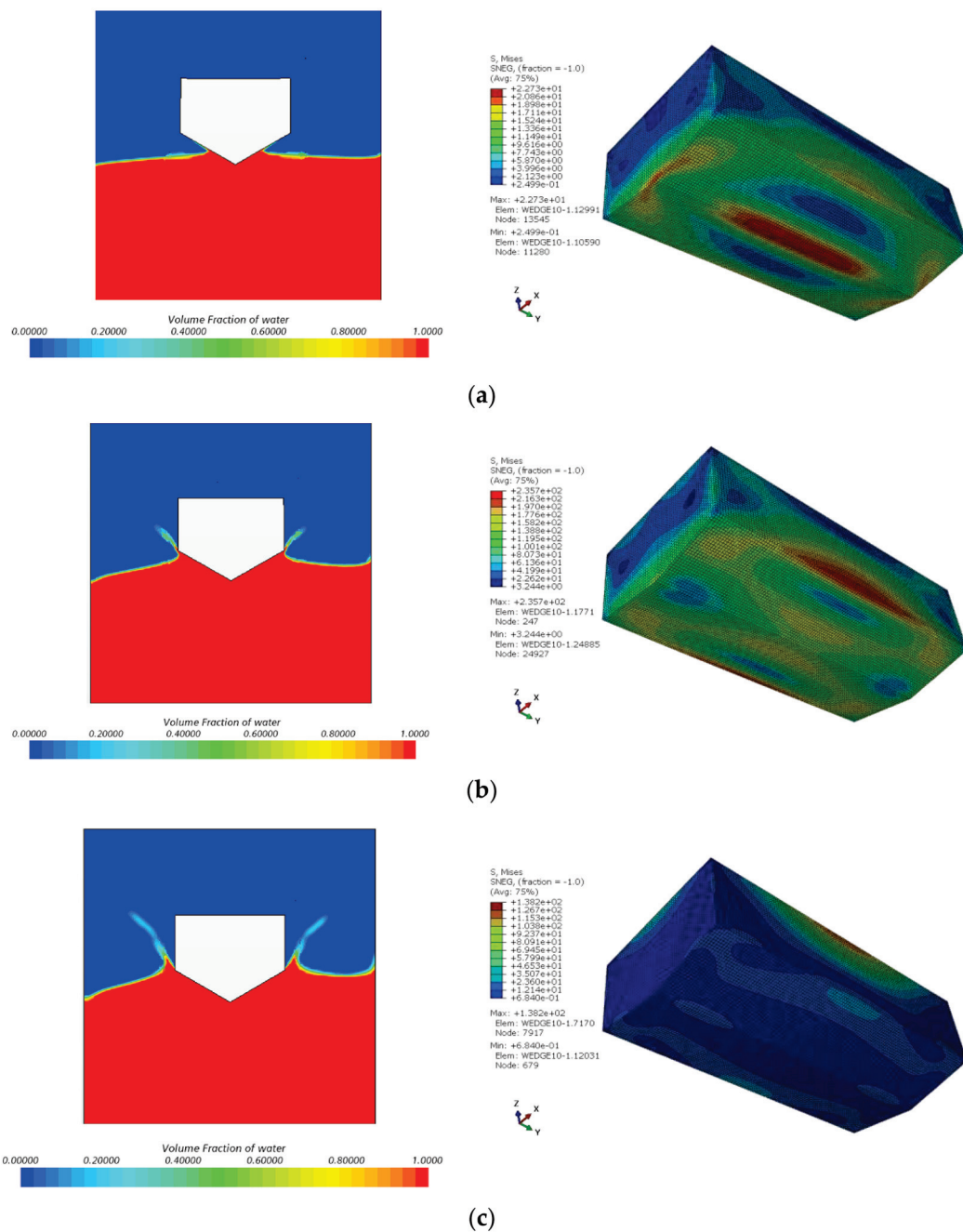


Figure 16. Stress nephogram of a wedge with $\beta = 10^\circ$ and a thickness of 5 mm at different times. (a) $t = 0.001$ s. (b) $t = 0.0022$ s. (c) $t = 0.0042$ s.

4.4. Effect of Structural Response for the Ship Model

The Shape of the Free Surface and the Pressure Cloud Diagram at the Moment of Impact

Figure 17 illustrates the situation before and after the slamming of the aluminum model ship in condition 1. When the vessel floated on the waves, it experienced pitching and heaving motions due to the periodic fluctuations of the waves. From the figure, it can be observed that the bow of the ship tended to emerge from the water when it was positioned in a wave trough, and a relatively large relative impact occurred when the next wave crest arrived, leading to a slamming phenomenon. The figure mainly presents the shapes of the free surface of the model at different times and the pressure cloud diagram on the bottom surface of the ship. Figure 17a shows the shape of the free surface and the pressure cloud diagram on the model surface at $t = 1.48$ s. At this moment, the wave crest

was located in the middle to aft position of the model, while the bow of the model was situated in a wave trough, with the bow surface completely emerging from the water. The bottom pressure was concentrated in the region of the bottom surface toward the stern, with the least pressure at the bow. Figure 17b indicates that at $t = 2.06$ s, the stern of the model was about to enter a wave trough, while the bow was positioned at the wave crest. The bow of the model was subjected to wave impact, and the bow surface of the model made full contact with the wave surface, with maximum pressure concentrated at the bow. Figure 17c shows that at $t = 2.5$ s, the wave crest had moved away from the bow toward the midship of the vessel, and the bow of the model gradually emerged from the water, with the maximum pressure concentrated in the front part of the ship's bottom. During this condition, there were no significant surface deformations or wave-breaking phenomena on the wave surface.

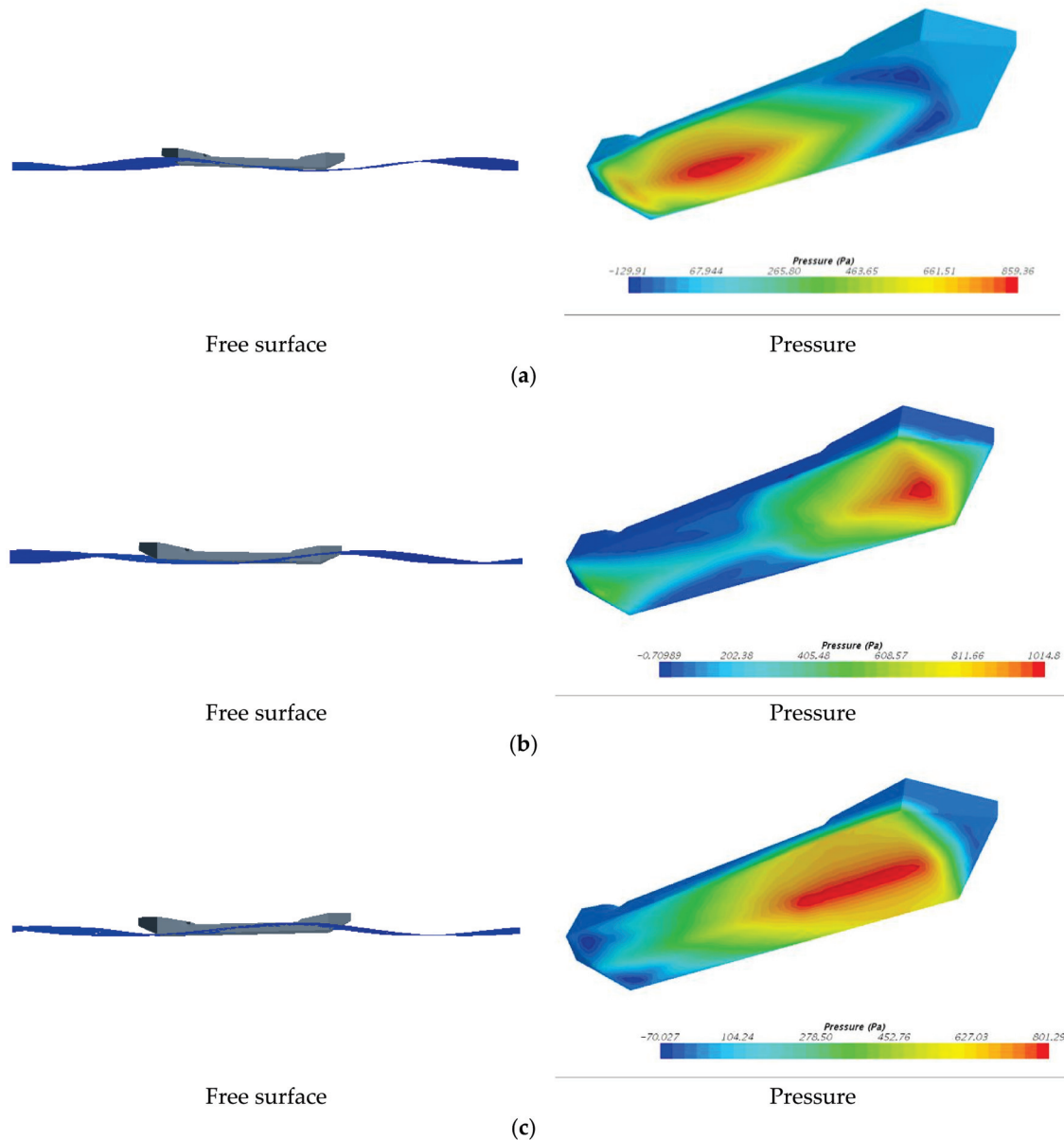


Figure 17. The free surface elevation and the pressure distribution on the ship model at different times in the case 1. (a) $t = 1.48$ s. (b) $t = 2.06$ s. (c) $t = 2.5$ s.

As shown in Figure 18, a series of measurement points were arranged on the bottom of the ship to detect the slamming pressure and structural deformation displacement experienced by the ship in the waves. Point P1 was located at the top end of the bow, with

coordinates (3.8, 0, 0.2). Points P2 and P3 were located at the bottom of the bow at the turning point, with coordinates (3.4, 0, 0) and (3.4, -0.2, 0.026), respectively. Point P4 was located at the bottom turning point of the stern, with coordinates (0.4, 0, 0).

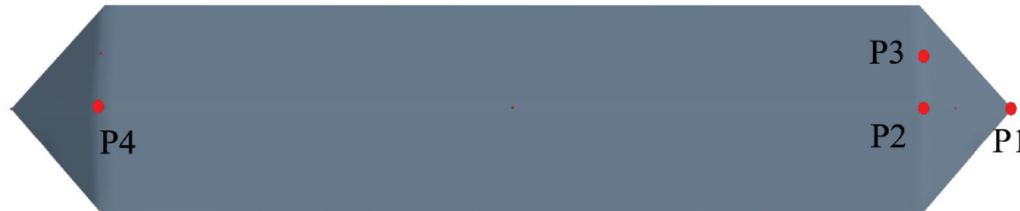


Figure 18. Schematic map of measuring point position.

Figure 19 displays the pressure and velocity time history curves for different measurement points during condition 1. The solid lines represent the slamming pressure curves, while the dashed lines represent the velocity curves. The left Y-axis represents pressure values, while the right Y-axis represents velocity values. From Figure 19, it can be observed that both pressure and velocity curves exhibited periodic variations, reflecting the frequent occurrence of slamming loads. Additionally, it can be noted from the figure that the pressure amplitude at point P1 (top end of the bow) was the smallest, at 0.314 kPa, while the maximum pressure amplitude at point P2 (bottom of the bow) was 1.64 kPa. At point P3, it was 0.884 kPa, and at point P4 (bottom of the stern), it was 0.90 kPa. Slamming phenomena occurred successively at the bow and stern of the ship. The pressure curve peaks in the figure show some attenuation, possibly due to the viscosity and damping effects caused by the turbulence model, leading to wave height attenuation. Moreover, the pressure curve at point P2 exhibited a sinusoidal function form, consistent with the input of regular waves. At this point, the peaks of the slamming pressure curve and velocity curve corresponded, indicating the correlation between slamming load and slamming velocity. The pressure amplitude and velocity amplitude at the remaining points had certain phase differences. Except for point P1, the pressure curves at the other points were relatively regular. However, the pressure curve at point P1 exhibited a typical abrupt change characteristic of slamming pressure, with a certain stable region, which is related to the position of point P1 at the top end of the bow.

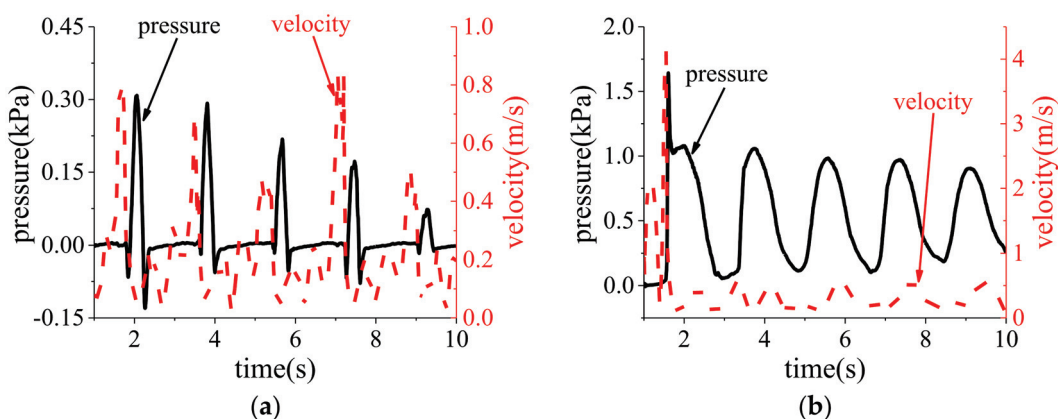


Figure 19. Cont.

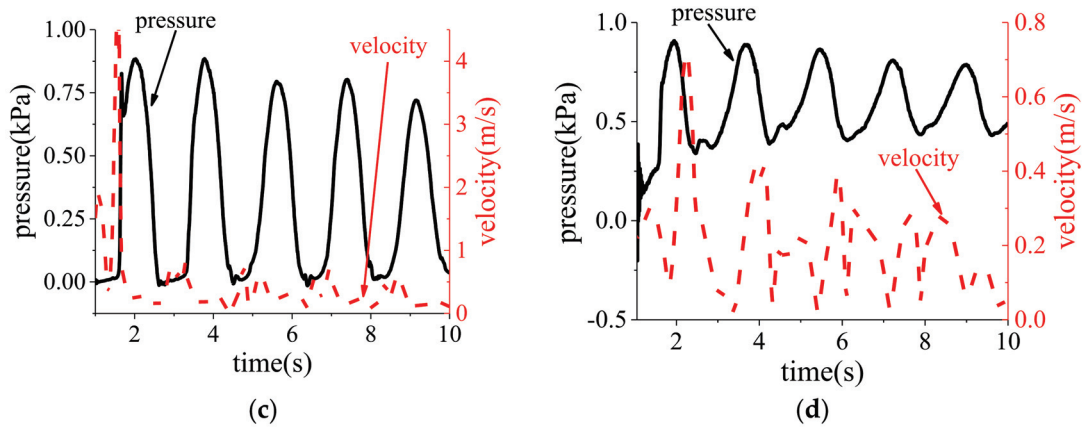


Figure 19. Correlation curve of pressure and velocity at measuring point in case 1. (a) P1. (b) P2. (c) P3. (d) P4.

Figure 20 shows the stress (unit: MPa) and strain cloud diagram of the model structure at different times in condition 1.

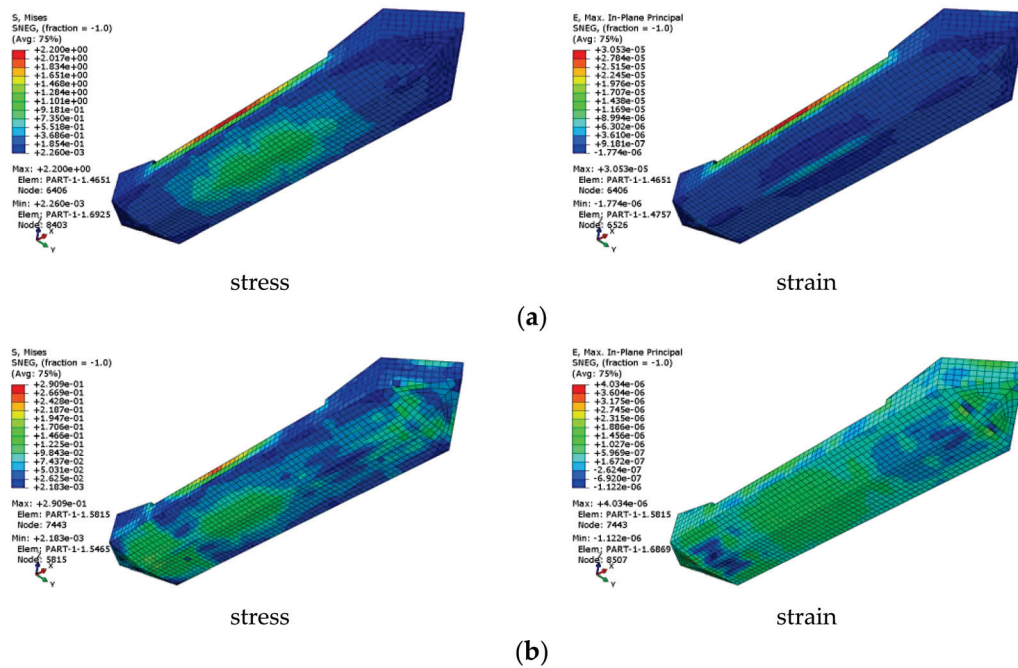


Figure 20. Stress and strain nephogram. (a) $t = 1.48$ s. (b) $t = 2.06$ s.

Figure 20a depicts the stress and strain cloud diagram of the model at $t = 1.48$ s. As indicated in Figure 17 earlier, at $t = 1.48$ s, the aft part of the hull was positioned at the crest of the wave, while the bow was fully emerged from the water, resulting in midship bulging. This corresponds to large stress and strain concentrations at the bottom and sides of the hull in Figure 20a, while stress and strain were relatively lower at the bow. Figure 20b presents the stress and strain cloud diagram of the model at $t = 2.06$ s. At this moment, the waves impacted the inclined part of the model's bow, leading to increased stress and strain in that region.

5. Conclusions

This study commenced with a series of controlled experiments, examining the impact of slamming on wedge-shaped bodies with varying inclinations from different heights. Following the empirical investigations, a comprehensive fluid–structure interaction (FSI)

numerical model was developed. This model integrated the dynamic capabilities of STAR-CCM+ for fluid dynamics with the structural analysis power of Abaqus, considering the effects of structural deformations. To validate the accuracy and reliability of the numerical model, its predictions were compared with both the outcomes of the initial wedge-shaped body water entry experiments and existing data from ship-slamming studies under wave conditions. The model demonstrated high efficacy in replicating observed phenomena. Detailed analyses based on the experimental data from wedge-shaped bodies explored how slamming pressures varied with both the inclination angle and the water entry velocity (corresponding to different falling heights). The study yielded critical insights into slamming loads, free surface dynamics, structural deformations, and stress–strain responses under fluid–structure interaction conditions. The specific conclusions are as follows:

- (1) The factors predominantly influencing slamming pressure are the inclination angle and water entry velocity, which exhibit distinct impacts under varying conditions:
 - (a) Inclination angle: For wedge-shaped bodies descending from a consistent height, an increase in the inclination angle results in a notable decrease in slamming pressure. At lower inclination angles, the dynamics of water surface lift-off and jet formation significantly alter both the location and intensity of the peak slamming pressure. In contrast, at higher inclination angles, the influences of lift-off and jetting on the peak slamming pressure are considerably diminished;
 - (b) Water entry velocity: For structures with a constant inclination angle that impact the water surface from varying heights, the increase in falling height leads to a corresponding rise in water entry velocity. This elevation in velocity systematically amplifies the slamming pressure exerted on the structure. This relationship underscores the critical role of water entry velocity in determining the slamming impact, necessitating precise measurement and consideration in the analysis of fluid–structure interactions during high-velocity water entries.
- (2) The correlation between simulated and experimental slamming pressures was strong at inclination angles of 30° and 40°, demonstrating a high degree of model fidelity in these scenarios. However, at the lower inclination angles of 10° and 20°, discrepancies emerged between the simulated and observed peak slamming pressures. This deviation suggests that the current simulation might not adequately capture the dynamics of thinner water jets formed at these angles. To enhance the accuracy of the simulations at lower inclination angles, a refinement of the computational mesh is recommended. This adjustment would enable more precise modeling of the jet phenomena and improve the alignment between simulated and experimental results.
- (3) The relationship between the inclination angle of a wedge-shaped body and the resulting slamming pressure, structural deformation, and vibration susceptibility is inversely proportional. Specifically, a smaller inclination angle leads to an increase in slamming pressure, which, in turn, causes more substantial structural deformation and heightens the likelihood of vibration. At these lower inclination angles, the position and intensity of the slamming pressure peak are significantly impacted by phenomena such as water surface lift-off and jet formation. This observation underscores the critical influence of hydrodynamic behaviors on the structural response during water entry events, necessitating detailed consideration in both experimental setups and numerical simulations to ensure accurate predictions and robust structural design.

Author Contributions: Conceptualization, T.L. and X.Z.; methodology, T.L.; software, T.L.; Writing—original draft, T.L.; Writing—review and editing, J.W. and K.L.; Visualization, J.W. and K.L. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Natural Science Foundation of China (Grant No. 52171311; Grant 52271279) and the Open Project of the National Key Laboratory of Waterway Traffic Control (Grant No. W23CG090243).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Military in Metal Processing. The Shame of Japan's Shipbuilding Industry-MOL COMFORT Shipbreaking Incident [EB/OL]. (2017-7-16). Available online: https://www.sohu.com/a/157626906_766672 (accessed on 3 March 2023).
2. Yang, B. Study on Dynamic Response and Ultimate Strength of Container Ship Hull Structure under Slamming Loads. Ph.D. Thesis, Shanghai Jiao Tong University, Shanghai, China, 2019.
3. Ochi, M. Prediction of slamming characteristics and hull responses for ship design. In Proceedings of the Annual Meeting of SNAME, New York, NY, USA, 15–17 November 1973.
4. Yu, P.; Ren, H.; Li, C.; Liu, Y.C. Direct Calculation of Design Loads for Ship Bow Slamming. *J. Ship Mech.* **2016**, *20*, 566–573. [CrossRef]
5. Hong, S.Y.; Kim, K.H.; Kim, B.W.; Kim, Y.S. Experimental Study on the Bow-Flare Slamming of a 10,000 TEU Containership. In Proceedings of the Twenty-Fourth International Ocean and Polar Engineering Conference, Busan, Republic of Korea, 15–20 June 2014.
6. Hermundstad, O.A.; Moan, T. Numerical and experimental analysis of bow flare slamming on a Ro-Ro vessel in regular oblique waves. *J. Mar. Sci. Technol.* **2005**, *10*, 105–122. [CrossRef]
7. Wang, S.; Guedes Soares, C. Experimental and numerical study on bottom slamming probability of a chemical tanker subjected to irregular waves. In *Maritime Technology and Engineering*; Taylor & Francis Group: London, UK, 2015; pp. 1065–1072.
8. Lindemann, T.; Backhaus, E.; Ulbertus, A.; Oksina, A.; Kaeding, P. Investigations on the dynamic collapse behaviour of thin-walled structures and plate panels for shipbuilding applications. In Proceedings of the International Ocean and Polar Engineering Conference, Big Island, HI, USA, 21–26 June 2015.
9. Wang, X.; Yang, P.; Gu, X.; Hu, J. Review of the theoretical investigation of slamming of global wave loads on ship structures. *J. Ship Res.* **2015**, *10*, 7–18.
10. Wang, X.; Gu, X.; Hu, J. Springing investigation of a ship based on model tests and 3D hydroelastic theory. *J. Ship Mech.* **2012**, *16*, 915–925.
11. Jiao, J.; Ren, H.; Sun, S.; Sun, L. Research on experimental technique of large-scale model under actual seaconditions. *Shipbuild. China* **2016**, *57*, 50–58.
12. Zhu, R.; Lu, J.; Ji, R.; Xia, M.; Li, L.; Han, Z. Numerical Simulation of Three-Dimensional Wedge Entry Slamming under Wave Action. *Ship Sci. Technol.* **2019**, *41*, 6–11.
13. Yang, B.; Wang, D. Dynamic Ultimate Hull Girder Strength Analysis on a Container Ship under Impact Bending Moments. *Int. J. Offshore Polar Eng.* **2018**, *28*, 105–111. [CrossRef]
14. Mackie, A. The water entry problem. *Q. J. Mech. Appl. Math.* **1969**, *22*, 1–17. [CrossRef]
15. Bilandi, R.N.; Jamei, S.; Roshan, F.; Azizi, M. Numerical simulation of vertical water impact of asymmetric wedges by using a finite volume method combined with a volume-of-fluid technique. *Ocean Eng.* **2018**, *160*, 119–131. [CrossRef]
16. Bilandi, R.N.; Dashtimanesh, A.; Mancini, S.; Vitiello, L. Comparative study of experimental and CFD results for stepped planing hulls. *Ocean Eng.* **2023**, *280*, 114887. [CrossRef]
17. Krastev, V.K.; Facci, A.L.; Ubertini, S. Asymmetric water impact of a two dimensional wedge: A systematic numerical study with transition to ventilating flow conditions. *Ocean. Eng.* **2018**, *147*, 386–398. [CrossRef]
18. Zhao, Z. Research on Prediction of Slamming Pressure Load on VLCC Based on Two-Step Method. Ph.D. Thesis, Tianjin University, Tianjin, China, 2016.
19. Zhang, B. Research on Strongly Nonlinear Slamming Loads of Ships Based on CFD Technology. Master's Thesis, Dalian Maritime University, Dalian, China, 2020.
20. Stavovy, A.B.; Chuang, S.-L. Analytical determination of slamming pressures for high-speed vehicles in waves. *J. Ship Res.* **1976**, *20*, 190–198. [CrossRef]
21. Peng, D. Research on Dynamic Response of Ship Hull Structure under Slamming Loads. Master's Thesis, Jiangsu University of Science and Technology, Zhenjiang, China, 2017.
22. Wang, J.; Zhao, X.; Liu, K.; Hong, Z.; Zhou, T.; Jina, M. Local Area Linear Optimization of Large Cruise Ships Based on Wave Slamming Loads. *Nav. Archit. Mar. Eng.* **2022**, *44*, 55–62+123. [CrossRef]
23. Chillemi, M.; Cucinotta, F.; Passeri, D.; Scappaticci, L.; Sfravara, F. *CFD-Driven Shape Optimization of a Racing Motorcycle*; Springer Nature: Cham, Switzerland, 2024.

24. Lee, E.J.; Diez, M.; Harrison, E.L.; Jiang, M.J.; Snyder, L.A.; Powers, A.M.R.; Bay, R.J.; Serani, A.; Nadal, M.L.; Kubina, E.R.; et al. Experimental and computational fluid-structure interaction analysis and optimization of Deep-V planing-hull grillage panels subject to slamming loads—Part II: Irregular waves. *Ocean Eng.* **2024**, *292*, 116346. [CrossRef]
25. Tuitman, J.T. Hydro-Elastic Response of Ship Structures to Slamming Induced Whipping. Ph.D. Thesis, Technische Universiteit Delft, Delft, The Netherlands, 2010.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

A Two-Step Approach for Evaluating the Dynamic Ultimate Load Capacity of Ship Structures

Weilong Zhang ^{1,2}, Jinju Cui ^{1,2,*} and Deyu Wang ^{1,2}

¹ State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200240, China; 20000926vl@sjtu.edu.cn (W.Z.); dywang@sjtu.edu.cn (D.W.)

² Institute of Marine Equipment, Shanghai Jiao Tong University, Shanghai 200240, China

* Correspondence: jinjucui@sjtu.edu.cn

Abstract: One important parameter for evaluating the safety and reliability of a ship is the dynamic ultimate load capacity of ship structures. Because of the importance of this parameter, its determination is essential. In this paper, a novel “two-step” approach for determining the dynamic ultimate load capacity of ship structures is proposed. The main idea of two-step approach is to determine the dynamic ultimate load capacity based on the static ultimate load capacity after accounting for impacts that cause strain on the ship structures. This approach is based on nonlinear finite element method. Here, taking stiffened plate as a case study, the practical application of this two-step approach is discussed in detail. The results of this approach reveal that the static ultimate load capacity decreases by less than 3% after a stiffened plate is subjected to an impact load whose amplitude corresponds to the dynamic ultimate load capacity. Then, the influence of the impact duration on the failure mode and the effect of the impact load cycles and the impact load sequence on the dynamic ultimate load capacity of the stiffened plate were investigated. Finally, the applicability of the two-step approach to a hull girder is demonstrated. The two-step approach and the conclusions presented in this paper can provide guidance for the evaluation of dynamic ultimate load capacity.

Keywords: dynamic ultimate load capacity; two-step approach; failure mode; ship structures; stiffened plate; impact load cycle; impact load sequence

1. Introduction

The ultimate load capacity of a ship is a crucial parameter for assessing its safety and reliability and has been extensively investigated by scholars. To date, numerous well-established methodologies have been developed to compute the static ultimate load capacity of ship structures. Smith [1] proposed an incremental-iteration method for calculating the ultimate load capacity of a hull girder. In this method, the cross-section is divided into different types of calculation units, each with its own specific stress–strain curve. By gradually increasing the curvature of the cross-section, calculating the stress of each unit, and integrating the stress of all units on the cross-section, one can obtain the bending moment–curvature curve where the peak value of the curve is the static ultimate load capacity of the hull girder. The curvature of the cross section refers to the angle at which the cross section of the hull girder rotates around the neutral axis. Because of its simplicity and high calculation accuracy, the incremental-iteration method has become a standard method for calculating the static ultimate load capacity of hull girders in classification societies. Tanaka et al. [2] extended the incremental-iteration method and established a method for calculating the ultimate load capacity of ship under combined bending and torsion, considering the influence of shear stress. Ueda and Sherif [3] proposed a numerical method called the Idealized Structural Unit Method (ISUM) for calculating the ultimate load capacity of ship structures. The structural units in the ISUM are larger than the elements in the finite element method (FEM), which greatly reduces the computation time [4]. Lindemann and

Kaeding [5] used the ISUM to study the ultimate load capacity of the stiffened plates under lateral pressure and in-plane compression. Underwood et al. [6] proposed a new ISUM that can assess the static ultimate load capacity of damaged structures through collapse analysis of the stiffened plates.

In recent years, with the rapid improvement in computer performance, the nonlinear FEM has been widely used to compute the ultimate load capacity of ship structures. The nonlinear FEM is characterized by changes in structural stiffness in the FE analysis. There are three kinds of nonlinearity: geometric nonlinearity, material nonlinearity and boundary nonlinearity. The term “geometric nonlinearity” mainly refers to large deformations. The term “material nonlinearity” mainly refers to the consideration of the yield strength of the material. The term “boundary nonlinearity” mainly refers to changes in boundary conditions encountered during the analysis. It is common to encounter both geometric nonlinearity and material nonlinearity when analyzing ultimate load capacity. Thus, FE analysis is nonlinear. The ultimate load capacity of ship structures can be predicted by empirical formulas based on the numerous results obtained by the FEM. Zhang and Khan [7] proposed a semianalytical empirical formula and then performed numerical simulation of stiffened plates using FEM to determine the coefficients of the empirical formula. Kim et al. [8] proposed an empirical formula to predict the ultimate load capacity of the stiffened plates subjected to longitudinal compression through numerical calculations of the stiffened plates with T-bar and flat-bar stiffeners. Xu et al. [9] studied the effects of lateral pressure and stiffener types on the collapse behavior of the stiffened plates through nonlinear FEM and derived an empirical formula for predicting the ultimate load capacity of the stiffened plates under the combined actions of axial compression and different levels of lateral pressure based on the results of the numerical calculation.

In all of the above methods, the ultimate load capacity of ship structures is evaluated under static or quasistatic conditions. However, when a ship encounters extreme conditions such as large freak waves or underwater bubbles, the amplitude of the load on the ship is very large and the load duration can be reduced to the order of milliseconds [10,11]. Obviously, the load on ship structures cannot be considered as a static or quasistatic load in such situations. In particular, the incident report from *MOL Comfort* indicates that dynamic loads may be one of the causes of ship structure collapse [12]. Thus, it is meaningful to study the dynamic ultimate load capacity of ship structures. Yamada [13] studied the effect of strain rate on the ultimate load capacity of container ships by using a half-sinusoidal dynamic load. The numerical calculation revealed that the strain rate has a significant effect on the ultimate load capacity and that the ultimate load capacity could be increased by 10–20% when the strain rate was considered. Jagite et al. [14] found that in calculating the dynamic ultimate load capacity of ship structures subjected to wave loads, the effect of strain rate can be ignored for container ships. Yang et al. [15] studied the dynamic ultimate load capacity of rectangular plates under axial compression. These authors used the FE results for numerous ship plates to derive an empirical formula for predicting the dynamic ultimate compressive capacity of ship plates. Yang and Wang [16] studied the dynamic buckling of the stiffened plates under in-plane impact loads by theoretical derivation and considered the influence of rotational constraint stiffness on the dynamic response of the stiffened plates. Paik [17] studied the dynamic ultimate load capacity of plates under axial compression loads with different loading speeds. The experimental results show that as loading speed increases, the ultimate compressive capacity of the plate also increases gradually. Liu et al. [18] studied the dynamic failure of rectangular plates under lateral impact via impact experiments and FE analysis. Jagite et al. [19] studied the dynamic ultimate load capacity of the stiffened plates under axial compression and lateral load in a parameterized way. Load values obtained by parameterization are more consistent with those measured under real-world conditions. The limitations of the existing strain rate model are also discussed.

The current method for determining the dynamic ultimate load capacity of ship structures relies primarily on the concept of dynamic buckling. Ari-Gur and Simonetta [20]

defined the dynamic failure criterion as the case in which a small increase in the dynamic load amplitude causes a sudden increase in the dynamic response of the structure. Xiong et al. [21] studied the dynamic ultimate compressive load capacity of the stiffened plates based on this criterion and derived a reasonably accurate empirical formula. Yang et al. [22] proposed a one-time thickness-deformation method for determining the dynamic ultimate load capacity by studying a container ship's bow. Their formula defines the dynamic ultimate load capacity as the amplitude of the dynamic load at which the maximum deflection of the structure is equal to one times plate thickness. Yang and Wang [16] proposed a new formula for determining the dynamic critical buckling load. According to this formula, when the slope of the load-displacement curve at any given point reaches 35, the corresponding load at that point is the dynamic critical buckling load. There are other methods for determining the dynamic ultimate load capacity, but they are similar to those already described and will not be discussed further here. However, these methods all essentially use the deformation of the structure to determine the dynamic ultimate load capacity. There are drawbacks to applying these methods in some cases, as will be described below. Therefore, we hope to develop a novel method for determining the dynamic ultimate load capacity of ship structures.

Considering the context described above, this study uses a nonlinear FEM to investigate the dynamic ultimate load capacity of the stiffened plates as one of the primary components of ship structures. This work proposes a novel "two-step" approach to determine the dynamic load capacity of the stiffened plates based on the static ultimate load capacity after impact. This method is theoretically applicable to a variety of ship structures. Subsequently, the failure mode of stiffened plates under impact load is discussed. Considering that real ship structures may experience continuous impacts with different load amplitudes, this work includes an analysis of the dynamic ultimate load capacity of the stiffened plates subjected to continuous impact load, as well as of the influence of the impact load cycles and impact load sequence on the dynamic ultimate load capacity of the stiffened plate. Finally, the applicability of the two-step approach to hull girders is studied.

2. Methodology

2.1. Two-Step Approach

The dynamic ultimate load capacity of the ship structure is usually determined by iterative methods, i.e., the duration of the dynamic load is fixed, and the maximum response of the structure is found by increasing the amplitude of the dynamic load. The response-amplitude curve is derived from these data, and the point on the curve where the slope changes fastest is the dynamic ultimate load capacity of the structure [21,23].

Jagite et al. [24] defined dynamic ultimate capacity as the maximum load that can be applied to the ship structure without causing structural collapse. When the impact duration is large, the dynamic ultimate load capacity becomes easier to calculate by the FEM. As the amplitude of the impact load increases, the structure begins to undergo large deformations (as shown in Figure 1a). The dynamic effect is not obvious at this time. However, when the duration of the impact load is relatively small, that is, when the impact duration is less than the natural vibration period of the structure, the response-amplitude curve of the structure is that shown in Figure 1b. Under those circumstances, the dynamic ultimate load capacity is difficult to determine.

This paper presents a novel "two-step" method for determining the dynamic ultimate load capacity of ship structures. The main idea of the two-step approach is to determine the dynamic ultimate load capacity of the ship structure by using the static ultimate load capacity after impact. After a ship structure is subjected to an impact load, the ship structure may exhibit plastic deformation. Too large a plastic deformation will reduce the static ultimate load capacity of the ship structure.

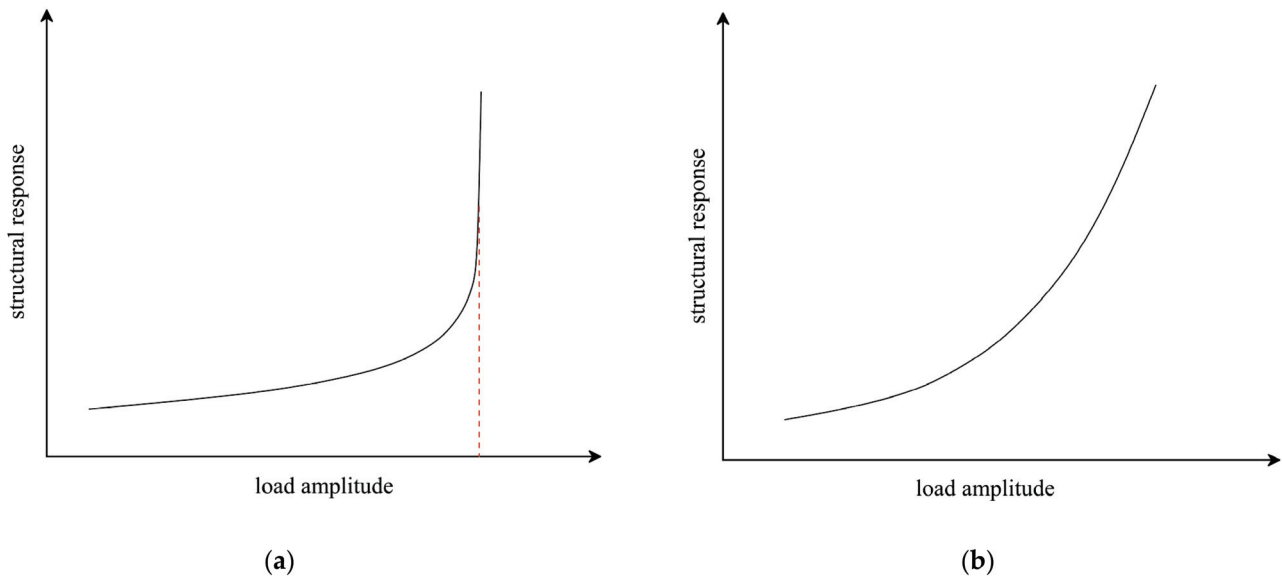


Figure 1. Structural dynamic response-impact load amplitude curve. (a) Curve for when the impact duration is large. (b) Curve for when the impact duration is relatively small.

Step 1 of the two-step approach involves calculating the dynamic response of the ship structure under impact load, and step 2 of the two-step approach involves calculating the static ultimate load capacity after impact. Step 1 and step 2 are continuous, and step 2 completely inherits the deformation and stress state of the structure in step 1. It should be noted that step 2 is carried out after the dynamic response of the structure in step 1 becomes stable. The schematic diagram in Figure 2 illustrates the two-step process, using a stiffened plate as an example. The static ultimate load capacity of the stiffened plate is denoted as σ_{u0} , and the static ultimate load capacity after impact is σ_{u_i} . In the application of the two-step approach, it is necessary to increase the amplitude of the dynamic load for continuous iterative calculation. In order to avoid including too many iterations, the static ultimate load capacity after impact is used to limit the number of iterations. In the process diagram, the static ultimate load capacity after impact cannot be less than 80% of the static ultimate load capacity, i.e., $\sigma_{u_i} \geq 0.8\sigma_{u0}$. After the calculation is completed, two curves can be obtained: the curve of the peak value of dynamic response with the dynamic load amplitude and the curve of the static ultimate load capacity after impact with the dynamic load amplitude. The two-step approach to determining the dynamic ultimate load capacity will be discussed in detail in the third section.

The two-step approach uses the nonlinear FEM to determine the dynamic ultimate load capacity of a ship structure. As a mature method, the FEM is often used for structural analysis. The FEM for calculating the ultimate load capacity of ship structures is known to other researchers [25,26]. The details of the FEM-based process for determining the dynamic ultimate load capacity of a ship structure by the two-step approach are shown in Figure 3. As a basic ship structure, stiffened plates are studied by many scholars. For this reason, the procedures of the two-step approach and determining the dynamic ultimate load capacity are here illustrated via an analysis of a stiffened plate. In order to demonstrate that the two-step approach can be used to determine the dynamic ultimate load capacity of other ship structures, a hull girder is also studied.

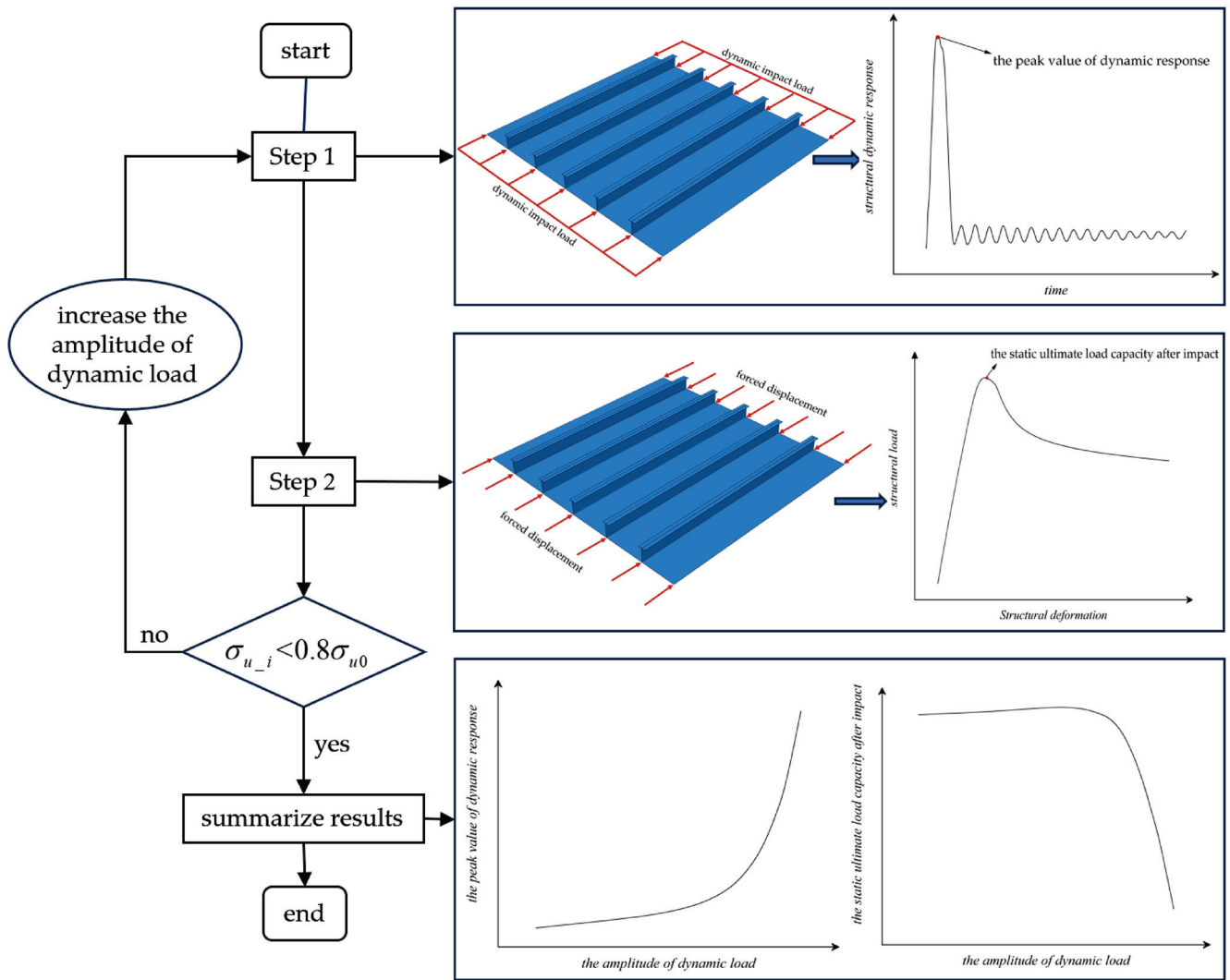


Figure 2. Schematic diagram of the two-step approach.

2.2. FE Modeling of a Stiffened Plate

2.2.1. Geometric Data

Figure 4 shows the geometric properties of the stiffened plate, where x , y and z correspond to the longitudinal, transverse and vertical directions, respectively. The ‘1/2 + 1 + 1/2 bay & 1/2 + 1 + 1/2 span’ model was used in this study. According to the recommendation of ISSC 2012, using the longitudinal range of ‘1/2 + 1 + 1/2 span’ can bring the boundary conditions of the FE model close to the boundary conditions of a stiffened plate on a real ship, making the results of FE calculation more accurate. The length, the width, the width between stiffeners and the thickness of the stiffened plate are denoted as a , B , b and t_p , respectively. A diagram of the cross-section of the T-bar stiffener is shown in Figure 5. The height and thickness of the stiffener’s web are denoted as h_w and t_w , respectively. The width and thickness of the stiffener’s flange are denoted as b_f and t_f , respectively. The geometric dimensions of the stiffened plate in this paper are shown in Table 1. Three different sizes of plate thickness and stiffeners were used. The plate slenderness ratio is $\beta = b/t_p \cdot \sqrt{\sigma_y/E}$, and the stiffener slenderness ratio is $\lambda = a/(\pi r) \cdot \sqrt{\sigma_y/E}$, where $r = \sqrt{I/A}$ is the radius of gyration of the stiffener with full-width (b) plating. A is the area of the cross-section of the stiffener with full-width plating, and I is moment of inertia of the stiffener with full-width plating. They are expressed as follows:

$$\begin{aligned}
 A &= bt_p + h_w t_w + b_f t_f \\
 z_0 &= \left(\frac{bt_p^2}{2} + h_w t_w \left(t_p + \frac{h_w}{2} \right) + b_f t_f \left(t_p + h_w + \frac{t_f}{2} \right) \right) / A \\
 I &= \frac{bt_p^3 + h_w^3 t_w + b_f t_f^3}{12} + h_w t_w \left(z_0 - t_p - \frac{h_w}{2} \right)^2 + b_f t_f \left(z_0 - t_p - h_w - \frac{t_f}{2} \right)^2 + bt_p \left(z_0 - \frac{t_p}{2} \right)^2
 \end{aligned} \tag{1}$$

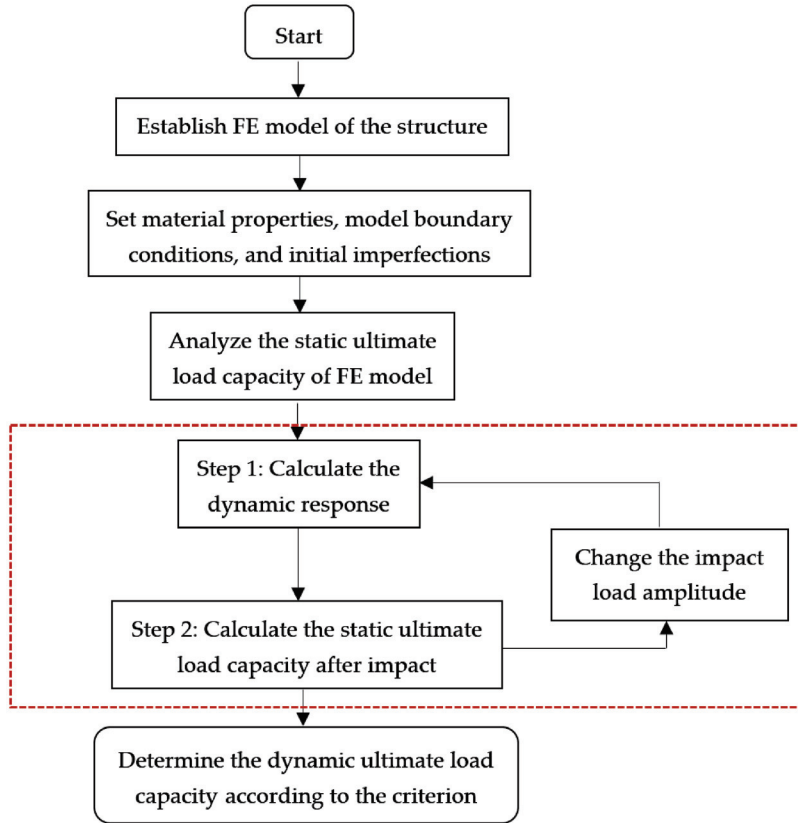


Figure 3. Flow chart of the FEM-based two-step approach to determining dynamic ultimate load capacity.

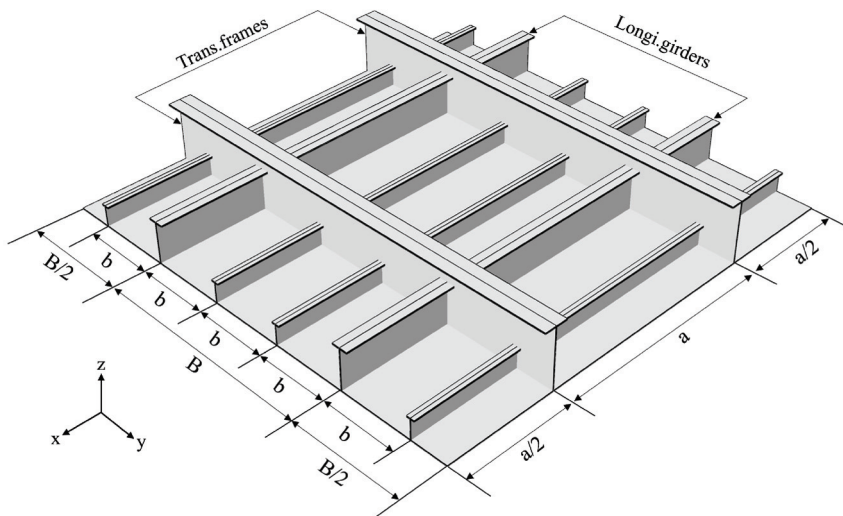


Figure 4. Geometric diagram of the stiffened plate.

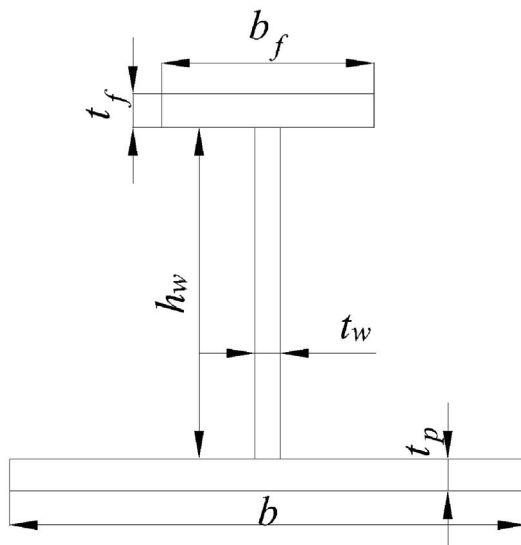


Figure 5. The cross-section of the stiffener.

Table 1. Geometric parameters of the stiffened plates.

Physical Quantity/Units	Symbol	Value
Plate length/mm	a	2550
Plate width/mm	b	850
Plate thickness/mm	t_p	11, 13, 16
Stiffener size 1/mm	$h_w \times b_f \times t_w/t_f$	$138 \times 90 \times 9/12$
Stiffener size 2/mm	$h_w \times b_f \times t_w/t_f$	$235 \times 90 \times 10/15$
Stiffener size 3/mm	$h_w \times b_f \times t_w/t_f$	$383 \times 100 \times 12/17$

2.2.2. Material Properties

The stiffened plates are made of high-tensile steel. The detailed material properties are as follows: Young's modulus $E = 2.06 \times 10^5$ MPa, Poisson's ratio $\nu = 0.3$, yield strength $\sigma_y = 313.6$ MPa and material density $\rho = 7.85 \times 10^{-9}$ t/mm³. The two-step approach requires calculation of the dynamic response of the stiffened plates and consideration of the effect of strain rate on the yield strength of the material. The influence of strain rate on yield strength of the material satisfies the Cowper–Symonds equation [27]. The Cowper–Symonds equation is as follows:

$$\sigma_{yd} = \sigma_y \left(1 + \left(\frac{\bar{\epsilon}}{D} \right)^{1/q} \right) \quad (2)$$

where σ_y is the static yield stress, σ_{yd} is the dynamic yield stress, $\bar{\epsilon}$ is the plastic strain rate, and D and q are constant parameters related to material properties. The effect of strain rates on materials was comprehensively summarized by Paik, based on a multitude of experimental results [28]. As the material used in this study is high-strength steel, the values of the constant parameters should be $D = 3200 \text{ s}^{-1}$ and $q = 5$.

2.2.3. Initial Imperfections

The initial imperfections caused by welding of the stiffened plate include deflection of the structure and welding residual stress, which reduce the ultimate load capacity of the structure. Because the residual stress has little effect on the ultimate load capacity of the stiffened plate [29–31], only the initial deflection of the stiffened plate was considered in the study. According to ISSC 2012 [32], three types of initial deflections were considered in this study: local deflection of the plate, overall deflection and tripping of the stiffener. The specific expressions are as follows:

The local deflection of plate, w_{pl} :

$$w_{pl} = A_0 \sin \frac{m\pi x}{a} \sin \frac{\pi y}{b} \quad (3)$$

where m is the number of longitudinal buckling half-wave of the local plate and m is the minimum integer that satisfies the following criterion:

$$\frac{a}{b} \leq \sqrt{m(m+1)} \quad (4)$$

A_0 is the deflection amplitude of the local plate. Smith et al. [33] summarized the amplitudes of initial deflection and divided them into three types:

$$A_0 = \begin{cases} 0.025\beta^2 t_p & \text{for slight level} \\ 0.1\beta^2 t_p & \text{for average level} \\ 0.3\beta^2 t_p & \text{for severe level} \end{cases} \quad (5)$$

The average level of initial deflection is used in this study.

The overall deflection, w_o :

$$w_o = B_0 \sin \frac{\pi x}{a} \sin \frac{\pi y}{B} \quad (6)$$

The tripping of the stiffener, w_s :

$$w_s = C_0 \frac{z}{h_w} \sin \frac{\pi x}{a} \quad (7)$$

where B_0 and C_0 are the amplitude of overall deflection and tripping of the stiffener, both of which were set to $0.0015a$.

The three types of initial deflection of the stiffened plate are shown in Figure 6.

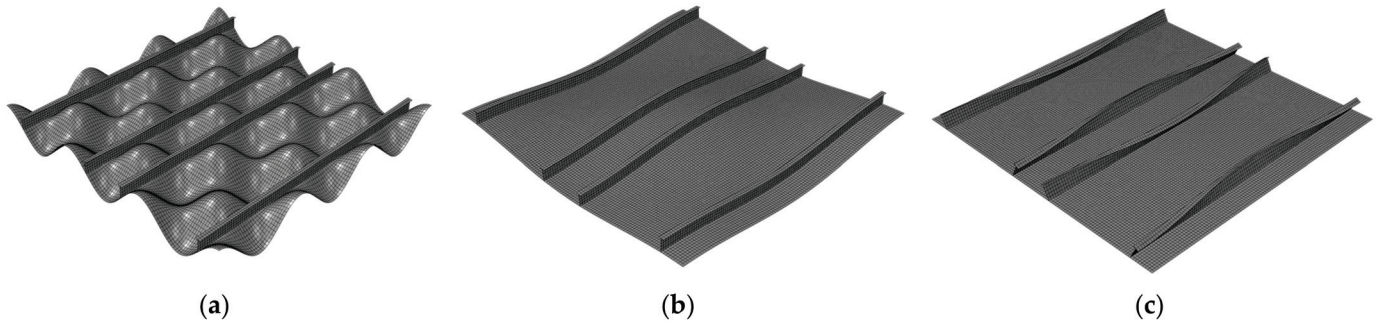


Figure 6. Initial geometric imperfections. (a) local deflection of plate. (b) overall deflection. (c) tripping of the stiffener.

2.2.4. Applied Loads and Boundary Conditions

The boundary conditions were based on ISSC 2012 [29]. The transverse frames and longitudinal girders were regarded as constraints and, for simplicity, were not included in the FE model of the stiffened plate, as shown in Figure 7. According to the function for the structure, the transverse frame intersection of C-C' and B-B' should restrict the z-direction displacement of plate nodes and the y-direction displacement of the stiffener nodes. The longitudinal girder intersection of A'-D' and A''-D'' should restrict the z-direction displacement of the plate nodes. The detailed boundary conditions of the stiffened plate are shown in Table 2.

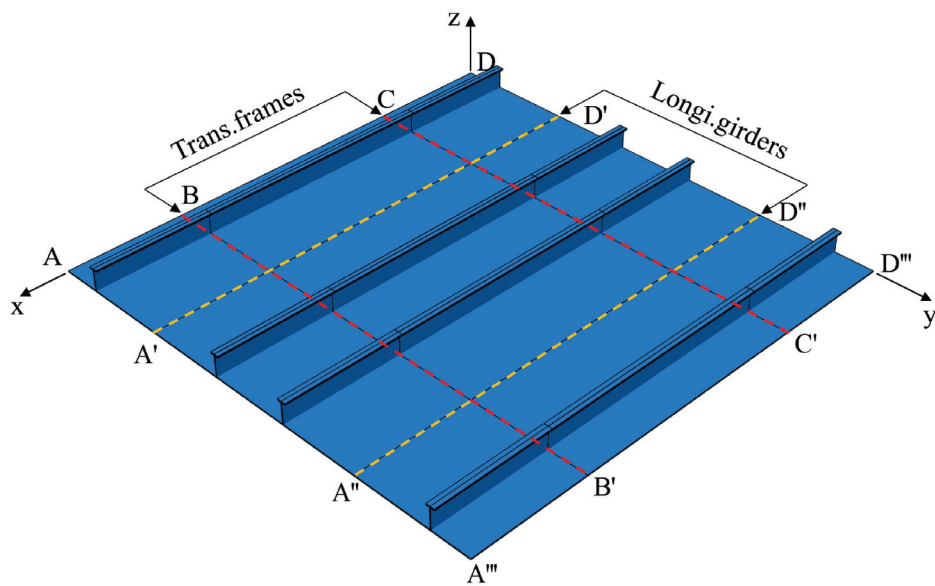


Figure 7. Schematic diagram of the simplified stiffened plate and coordinate system.

Table 2. Boundary conditions.

Location	Boundary Conditions
A – A''' and D – D'''	$R_y = R_z = 0$ and $U_x = \text{uniform}$
A – D and A''' – D'''	$R_x = R_z = 0$ and $U_y = \text{uniform}$
A' – D', A'' – D'', B – B' and C – C'	$U_z = 0$ at plate nodes, $U_y = 0$ at stiffener nodes

The axial impact load can be simplified as a force load, and it can be described by a half-sinusoidal function. The amplitude curve of the impact load is given by

$$P(t) = P_{\max} \sin\left(\frac{\pi t}{T_d}\right) \quad (8)$$

where $P_{\max}$ is the maximum value of the impact load and T_d is the duration of the impact load. The curve of the impact load is shown in Figure 8.

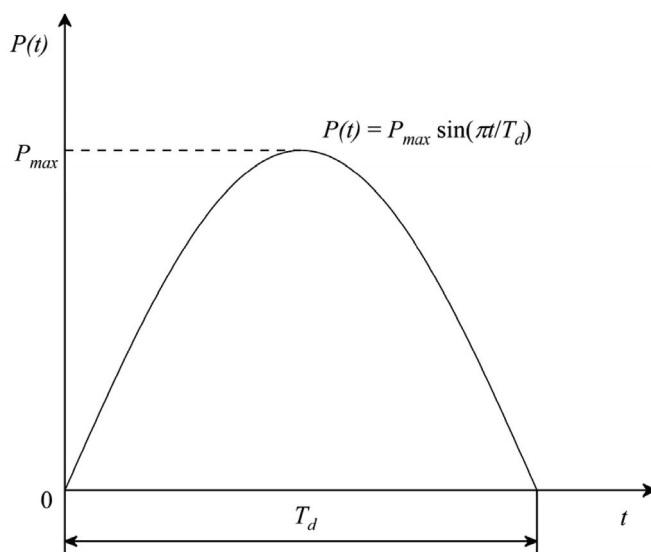


Figure 8. The curve of the impact load over time.

2.2.5. Non-Linear FE Modeling

The numerical calculations in this study were conducted in Abaqus 2021. In the process of numerical calculation, Dynamic-Implicit was used in step 1, and Static-General was used in step 2. The element type of FE model is S4R, which is a four-node, double-curved shell. Each node of the element has six degrees of freedom. An FE model of a typical stiffened plate is shown in Figure 9. The nodes at both ends are connected with the corresponding reference points by equation constraints to ensure that the displacements of the reference points and the nodes at both ends are equal in the x direction. In step 1, dynamic impact loads were applied to the reference points; in step 2, forced displacements were applied to the reference points.

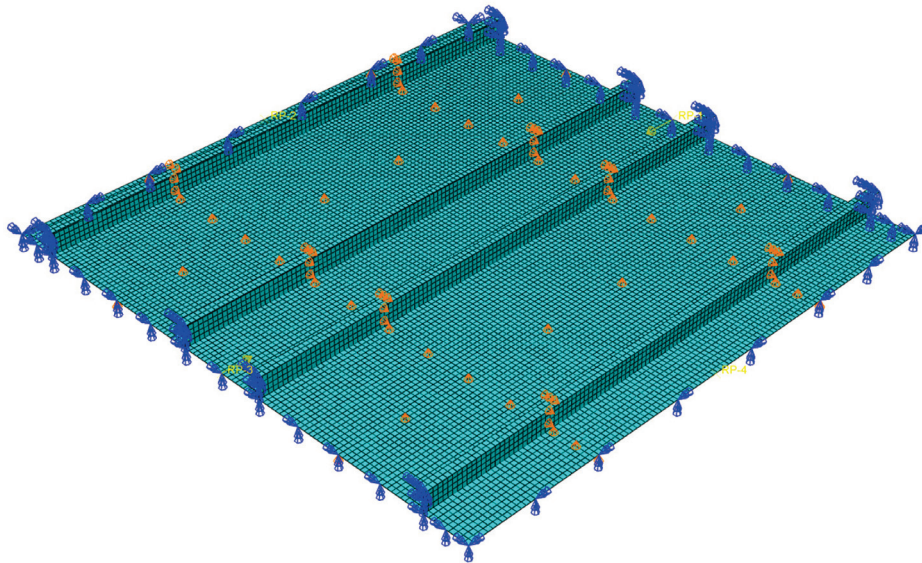


Figure 9. FE model of a typical stiffened plate.

3. Numerical Results and Discussion

The purpose of this study is to propose a new method for determining the dynamic ultimate load capacity of ship structures. In order to make this method more universal, five different sizes of the stiffened panels were designed according to Table 1, as shown in Table 3. Zhang and Khan [34] found that the plate slenderness ratio β ranges from 1.0 to 4.5 and that the stiffener slenderness ratio λ ranges from 0.15 to 0.95 on ship structures. As shown in Table 3, the range of λ and β in the designed model are both within the range reported by Zhang and Khan.

Table 3. Calculation model parameters.

Model	Thickness of the Plate (mm)	Size of the Stiffener	The Plate Slenderness Ratio β	The Stiffener Slenderness Ratio λ
Model 1	11	Size 1	3.02	0.65
Model 2	11	Size 2	3.02	0.36
Model 3	11	Size 3	3.02	0.21
Model 4	13	Size 2	2.55	0.38
Model 5	16	Size 2	2.07	0.40

3.1. Static Ultimate Load Capacity Results

Before calculating the dynamic ultimate load capacity, the static ultimate load capacity and the first-order vibration period of each model were calculated first. The impact load is determined by the impact duration and the impact load amplitude. Therefore, the static ultimate load capacity σ_{u0} and the first-order vibration period T_0 can be used as the

reference values for the amplitude of the impact load and the impact duration, respectively. Then, the nondimensional impact load amplitude and the nondimensional impact duration can be defined as $P_{\max}/\sigma_{u0}$ and T_d/T_0 , respectively.

The accuracy of FEM can be verified by comparing the FE results with those of empirical formula, as discussed by Xiong et al. [21]. The results for static ultimate load capacity were verified by comparison with the empirical formula derived by Xu et al. [9]. This formula is relatively accurate for predicting the static ultimate load capacity of T-bar stiffened plates. The empirical formula is as follows:

$$\frac{\sigma_u}{\sigma_y} = \frac{1}{\sqrt{\left(3.555 - 3.577\lambda - 3.424\beta + 0.999\lambda\beta + 4.373\lambda^2 + 1.812\beta^2\right) - 0.22\lambda^2\beta^2 - 2.584\lambda^3 - 0.277\beta^3 + 0.017\lambda^3\beta^3 + 0.458\lambda^4}} \quad (9)$$

The results for static ultimate load capacity of FEM, along with the empirical formula and errors, are shown in Table 4. All error values are within 10%. The maximum error is 6.41%. Xiong et al. [21] compared the static ultimate load capacity obtained by FEM and the empirical formula. In that study, the maximum error was 7.14%, and all errors were within 10%. Thus, the results of the FEM calculation in this study are sufficiently precise.

Table 4. The results for static ultimate load capacity from FEM and the empirical formula, with errors.

Model	FEM	Empirical Formula	Error
Model 1	0.61	0.65	5.84%
Model 2	0.64	0.68	5.08%
Model 3	0.67	0.69	3.35%
Model 4	0.67	0.71	4.96%
Model 5	0.73	0.78	6.41%

3.2. The Determination Criterion of the Dynamic Ultimate Load Capacity

The dynamic ultimate load capacity of each stiffened plate model was computed based on the first vibration period, considering 5 different impact durations: $0.2T_0$, $0.5T_0$, T_0 , $2T_0$ and $5T_0$. According to the calculation procedure of the two-step approach, the maximum value of the dynamic response of the stiffened plate was recorded in step 1, and then the static ultimate load capacity after impact was recorded in step 2. The peak value of the dynamic response is the maximum axial displacement of the loaded edge, i.e., $u_{\max}$. The maximum axial displacement of the stiffened plate under static ultimate load capacity is u_{static} . Thus, the nondimensional maximum axial displacement is defined as $u_{\max}/u_{\text{static}}$. The static ultimate load capacity of the stiffened plate after impact is denoted as σ_{u_i} . The nondimensional static ultimate load capacity after impact is defined as σ_{u_i}/σ_{u0} .

Model 3 was taken as an example to illustrate the calculation process in the two-step approach. The process of applying the two-step approach is similar to that used for other models. When the impact duration $T_d = T_0$, the results obtained by using the two-step approach are shown in Figure 10. The axial displacement of the stiffened plate over time subjected to impact loads with different amplitudes in step 1 is shown in Figure 10a. The axial displacement of the loaded edge can also be called “end shortening”. As shown in the figure, the stiffened plate vibrates after impact and then tends to stabilize. As load amplitude increases, the maximum axial displacement also increases gradually. When $P_{\max}/\sigma_{u0} = 1.0$, the stiffened plate undergoes almost no plastic deformation after impact. However, when $P_{\max}/\sigma_{u0} > 1.0$, the stiffened plate undergoes plastic deformation, and as the impact load amplitude increases, the plastic deformation of the stiffened plate increases gradually.

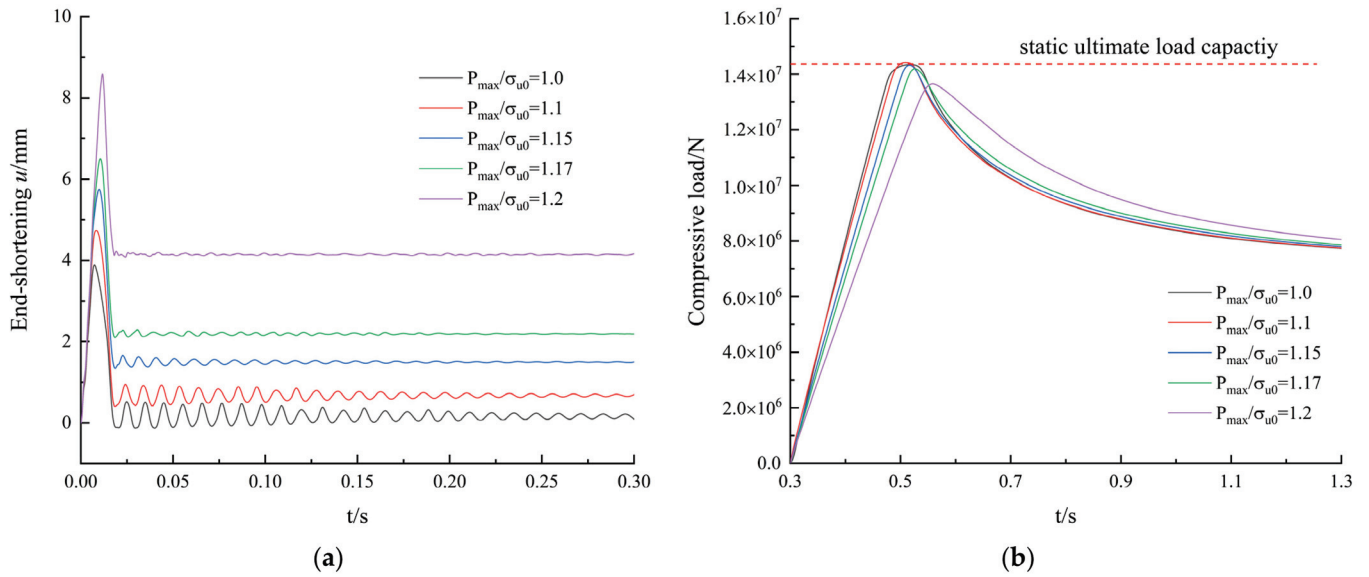


Figure 10. Calculation results from the two-step approach for Model 3. (a) The axial displacement over time under impact loads of different amplitudes with $T_d/T_0 = 1.0$ in step 1 (b) The compressive load over time after impacts of different amplitudes in step 2.

When the vibration of the stiffened plate becomes stable, the calculation in step 1 ends. The calculation in step 2 then begins, and forced displacement is applied at both ends of the stiffened plate to calculate the static ultimate load capacity. Figure 10b shows the compression load of the stiffened plate over time after impacts of different amplitudes. Note that step 1 was carried out in Dynamic-Implicit, with a total calculation time of 0.3 s, which is the real time. By contrast, step 2 was carried out in Static-General; in this step, the time is not real and defaults to 1 s. The maximum compressive load is the static ultimate load capacity after impact. It shown in Figure 10b that when $P_{\max}/\sigma_{u0} = 1.0, 1.1, 1.15$, the static ultimate load capacity after impact $\sigma_{u,i}$ does not decrease and is equal to the static ultimate load capacity σ_{u0} . However, when $P_{\max}/\sigma_{u0} = 1.1, 1.15$, the structure undergoes plastic deformation, which indicates that low levels of plastic deformation have no influence on the static ultimate load capacity of the stiffened plates. As the impact load amplitude increases, the static ultimate load capacity after impact decreases obviously at $P_{\max}/\sigma_{u0} = 1.2$.

The curve of the nondimensional maximum axial displacement versus nondimensional load amplitude and the curve of nondimensional static ultimate load capacity after impact versus nondimensional load amplitude from Model 3 can be obtained by gradually, iteratively increasing the amplitude of the impact load for each impact duration t . The results are shown in Figure 11.

By observing the trend in the maximum axial displacement versus the impact load amplitude in Figure 11a, we find that the maximum axial displacement gradually increases as the impact load amplitude increases and that the slope of the curve increases continuously. However, it shown in Figure 11b that the static ultimate load capacity after impact eventually stabilizes as the impact load amplitude increases, i.e., the static ultimate load capacity will not decrease after impact. Then, as the amplitude of the impact load continues to increase, the static ultimate load capacity after impact begins to decrease. The slope of the curve in Figure 11b is close to zero at the beginning, then begins to decrease; eventually, the slope becomes almost constant.

Therefore, we can find a point on the curve after which the slope of the curve remains constant, i.e., where the second derivative is 0, and define $P_{\max}/\sigma_{u0}$ of the point as the dynamic ultimate load capacity. The dynamic ultimate load capacity is denoted as σ_{ud} .

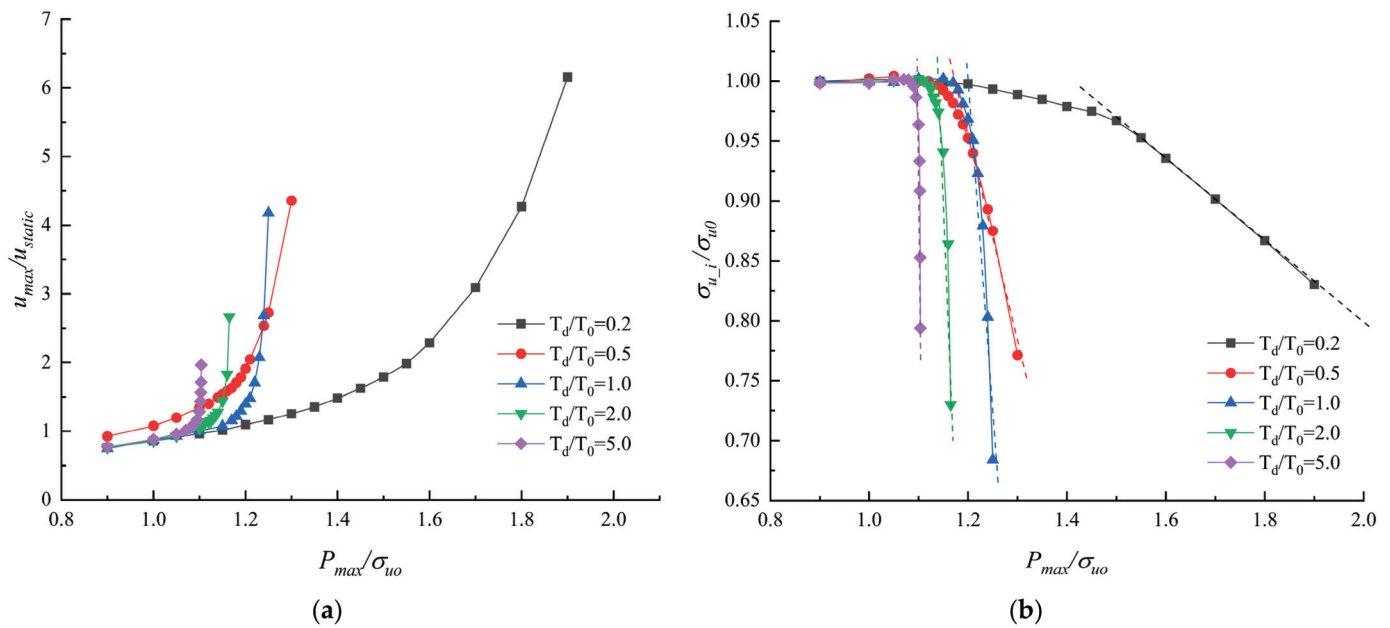


Figure 11. Calculation results from Model 3. (a) The nondimensional maximum axial displacement versus nondimensional load amplitude under impacts of different durations. (b) The nondimensional static ultimate load capacity after impact versus nondimensional load amplitude.

According to the above-described method of determining dynamic ultimate load capacity, the dynamic ultimate load capacity of Model 3 under different impact durations can be obtained, as shown in Table 5. The percentage reduction of static ultimate load capacity after impact is given in the table. After the stiffened plate is subjected to the impact load whose amplitude is the dynamic ultimate load capacity, its static ultimate load capacity is reduced by 1.4–2.9%.

Table 5. Dynamic ultimate load capacity and percentage reduction in static ultimate load capacity after impacts of different durations in Model 3.

T_d/T_0	σ_{ud}/σ_{u0}	$(\sigma_{u_i} - \sigma_{u0})/\sigma_{u0}$
0.2	1.5	2.9%
0.5	1.18	1.9%
1.0	1.19	2.8%
2.0	1.14	2.6%
5.0	1.095	1.4%

3.3. Dynamic Ultimate Load Capacity of Stiffened Plates

According to the procedure of the two-step approach, the calculation results from the other models are shown in Figure 12. The dynamic ultimate load capacity of each model under different impact durations is shown in Figure 13. As the impact duration increases, the dynamic ultimate load capacity of the stiffened plate gradually decreases. By observing the calculation results of Model 1 in Figure 13, we find that the dynamic ultimate load capacity is obviously smaller when $T_d/T_0 = 0.5$. Moreover, we observe the curve of the nondimensional maximum axial displacement versus nondimensional load amplitude of the stiffened plate in Figure 12a, and we find that the nondimensional maximum axial displacement corresponding to $T_d/T_0 = 0.5$ is the largest when $P_{max}/\sigma_{u0} < 1.1$. The cause of this phenomenon may be that higher-order deformation modes are activated under short impact duration. Figure 14 shows the displacement distribution at dynamic ultimate state of Model 1 with $T_d = 0.5T_0$ and the vibration mode with 103.36 Hz. As shown in Figure 14, the displacement distribution and vibration mode of the stiffened plate are similar. And

the vibration period is about $0.48T_0$ when frequency is equal to 103.36 Hz, which is close to the impact duration $T_d = 0.5T_0$. Therefore, when $T_d/T_0 = 0.5$, the impact load activates the high-order vibration mode of the stiffened plate, causing the resonance of the stiffened plate, thus the axial displacement of the stiffened plate increases significantly.

Table 6 shows the percentage reduction of static ultimate load capacity after impacts of different duration. The static ultimate load capacity of after impact all models is reduced by no more than 3%. It indicates that the static ultimate load capacity of the stiffened plate will decrease slightly when the stiffened plate is subjected to the impact load whose amplitude is the dynamic ultimate load capacity. We propose another method to determine the dynamic ultimate load capacity, that is, the static ultimate load capacity is reduced by 3% after the structure is subjected to the impact load, then the amplitude of this impact load is the dynamic ultimate load capacity. This method is suitable for the case where ship structures are subjected to impact loads of complex shapes.

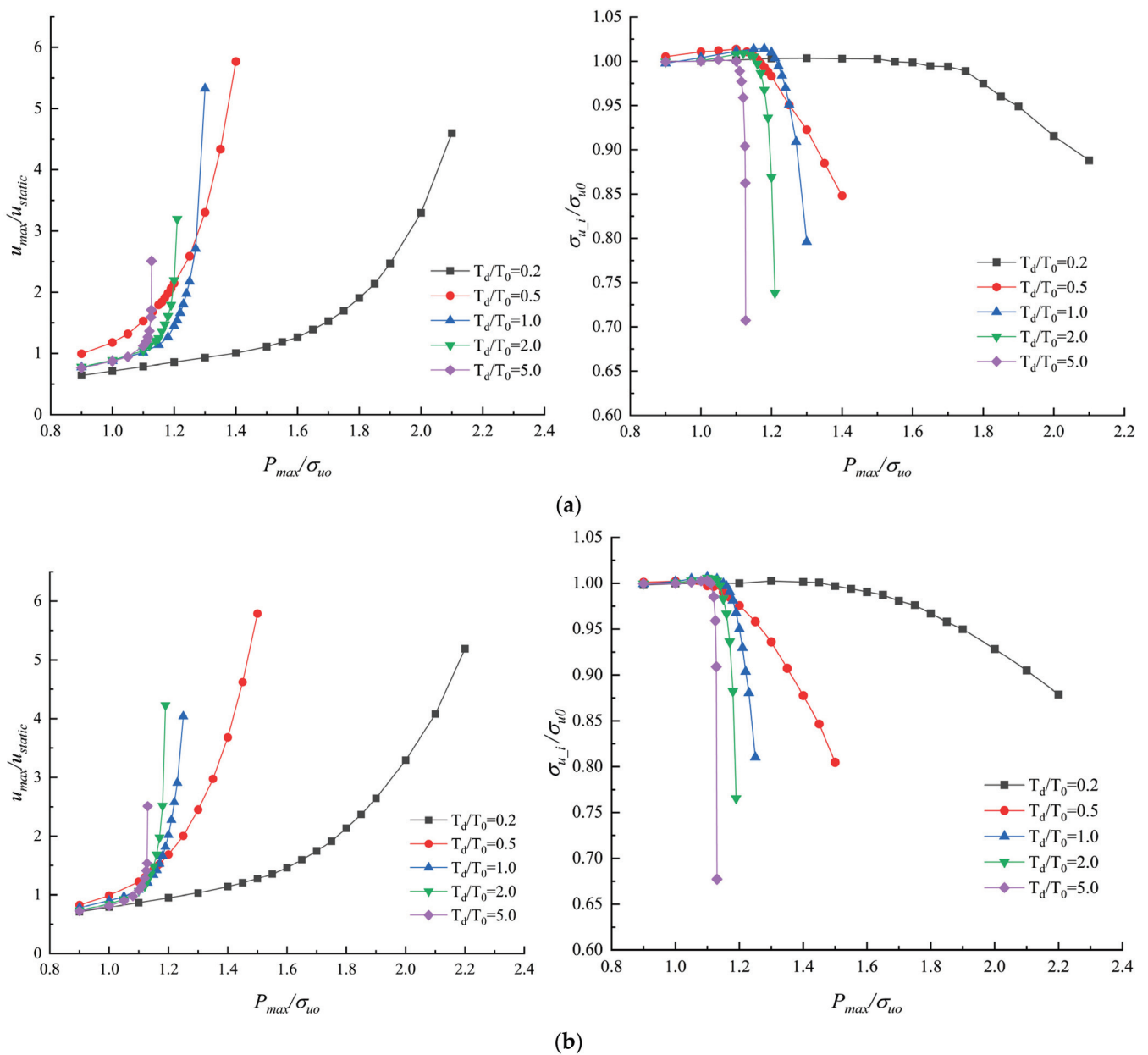


Figure 12. Cont.

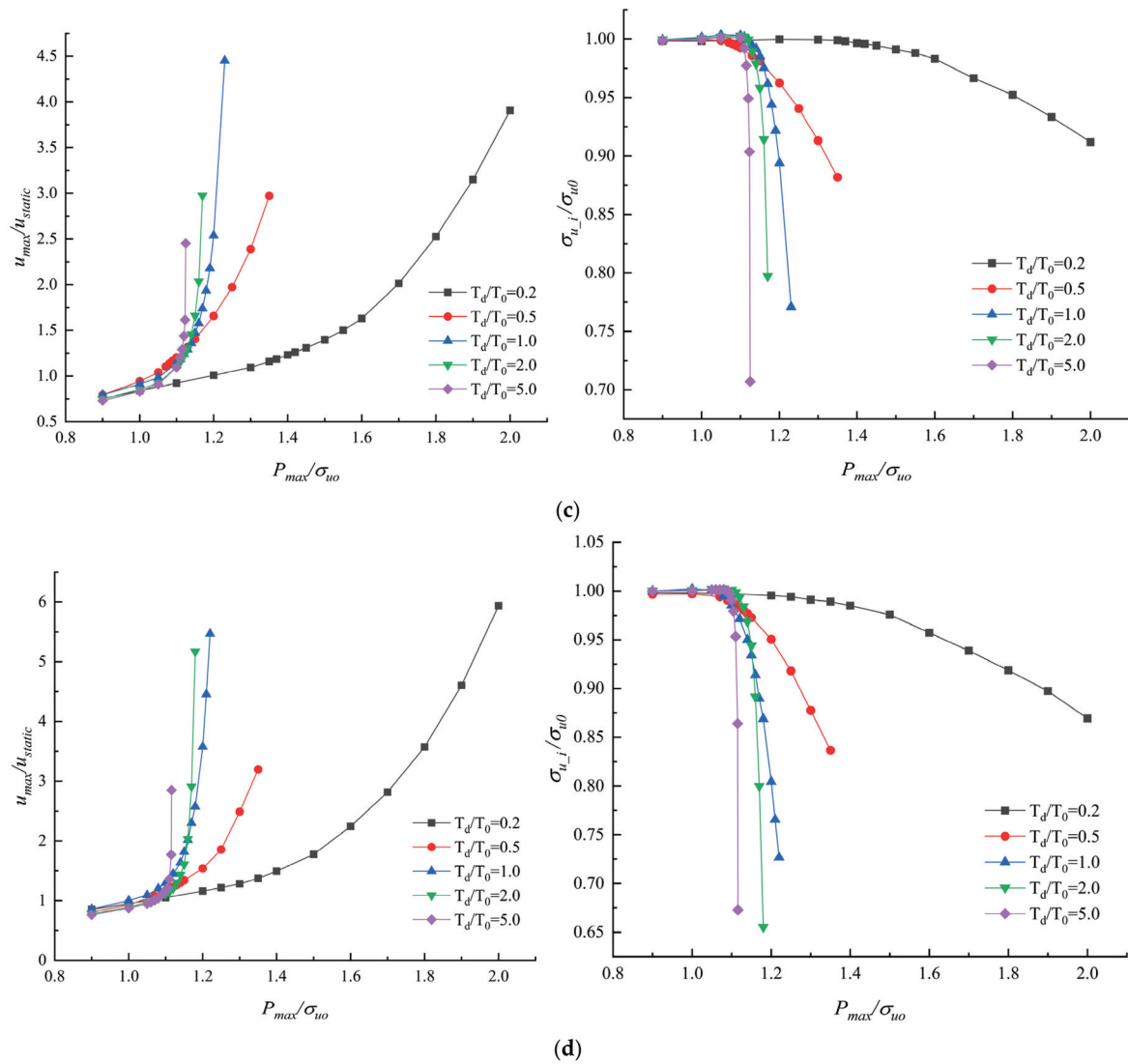


Figure 12. Calculation results from the two-step approach. (a) Model 1. (b) Model 2. (c) Model 4. (d) Model 5.

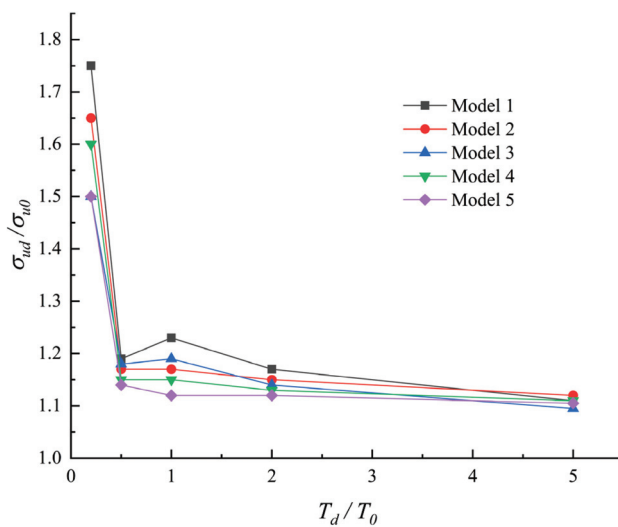


Figure 13. The curve of nondimensional dynamic ultimate load capacity versus nondimensional impact duration.

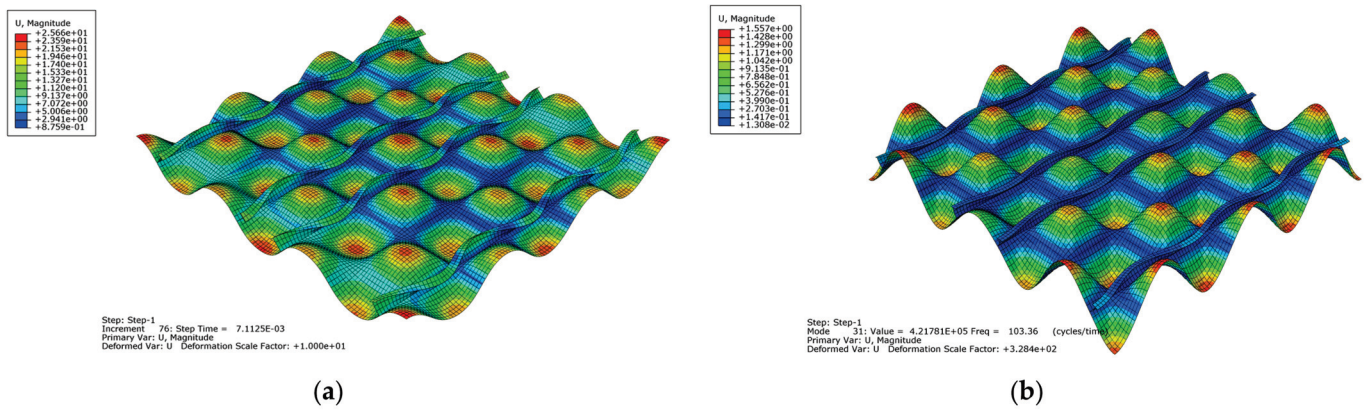


Figure 14. (a) Displacement distribution diagram for the dynamic ultimate state of Model 1 when $T_d/T_0 = 0.5$. (b) The vibration mode of Model 1 with 103.36 Hz.

Table 6. The percentage reduction of static ultimate load capacity after impacts of different durations.

T_d/T_0	Model 1	Model 2	Model 3	Model 4	Model 5
0.2	2.1%	1.7%	2.9%	1.7%	2.4%
0.5	1.2%	1.4%	1.9%	1.7%	2.4%
1	1.6%	1.0%	2.8%	1.6%	2.9%
2	1.4%	1.7%	2.6%	2.0%	1.6%
5	1.1%	1.5%	1.4%	1.0%	2.1%

3.4. Dynamic Failure Mode of Stiffened Plates

The failure mode of the stiffened plate can be divided into 6 types [34], as follows: mode I, overall failure mode; mode II, failure of the local plate between stiffeners; mode III, beam-column-type failure of the stiffeners with the attached plate; mode IV, local buckling of the stiffener web; mode V, tripping of the stiffener; mode VI, overall yielding of the stiffened plate. When stiffened plates are subjected to dynamic impact loads, the failure mode will change.

Figure 15 shows the stress diagram of Model 1 in the ultimate state. As shown in Figure 15a–e, the failure mode of the stiffened plate at the dynamic ultimate state changes as the impact duration increases. The failure mode of the stiffened plate is mainly failure of the local plate (mode II) when $T_d/T_0 = 0.2$. The deformation of the stiffener is relatively small. As the impact duration increases, both the tripping of the stiffener and the failure mode of the local plate arise when $T_d/T_0 = 0.5, 1.0$, i.e., the failure modes are mainly mode II and mode V. The beam-column-type failure of the stiffened plate (mode III) is the main failure mode when $T_d/T_0 = 5.0$. In addition, the failure mode at this time is the same as the static failure mode of the stiffened plate in Figure 15f.

Figure 16 shows the stress diagram of Model 3 in the ultimate state. It can also be seen from Figure 16a–e that as the dynamic impact duration increases, the failure mode of the stiffened plate changes. The failure mode of the stiffened plate is mainly failure of local plate (mode II) when $T_d/T_0 = 0.2$. The deformation of the stiffener is very small. when $T_d/T_0 = 0.5, 1.0$, the stiffener deformation increases, several half waves appear and the main failure modes are mode II and mode V. When $T_d/T_0 = 2.0, 5.0$, the tripping of the stiffener (mode V) is the main failure mode. The failure mode in $T_d/T_0 = 5.0$ is the same as the static failure mode of the stiffened plate in Figure 16f.

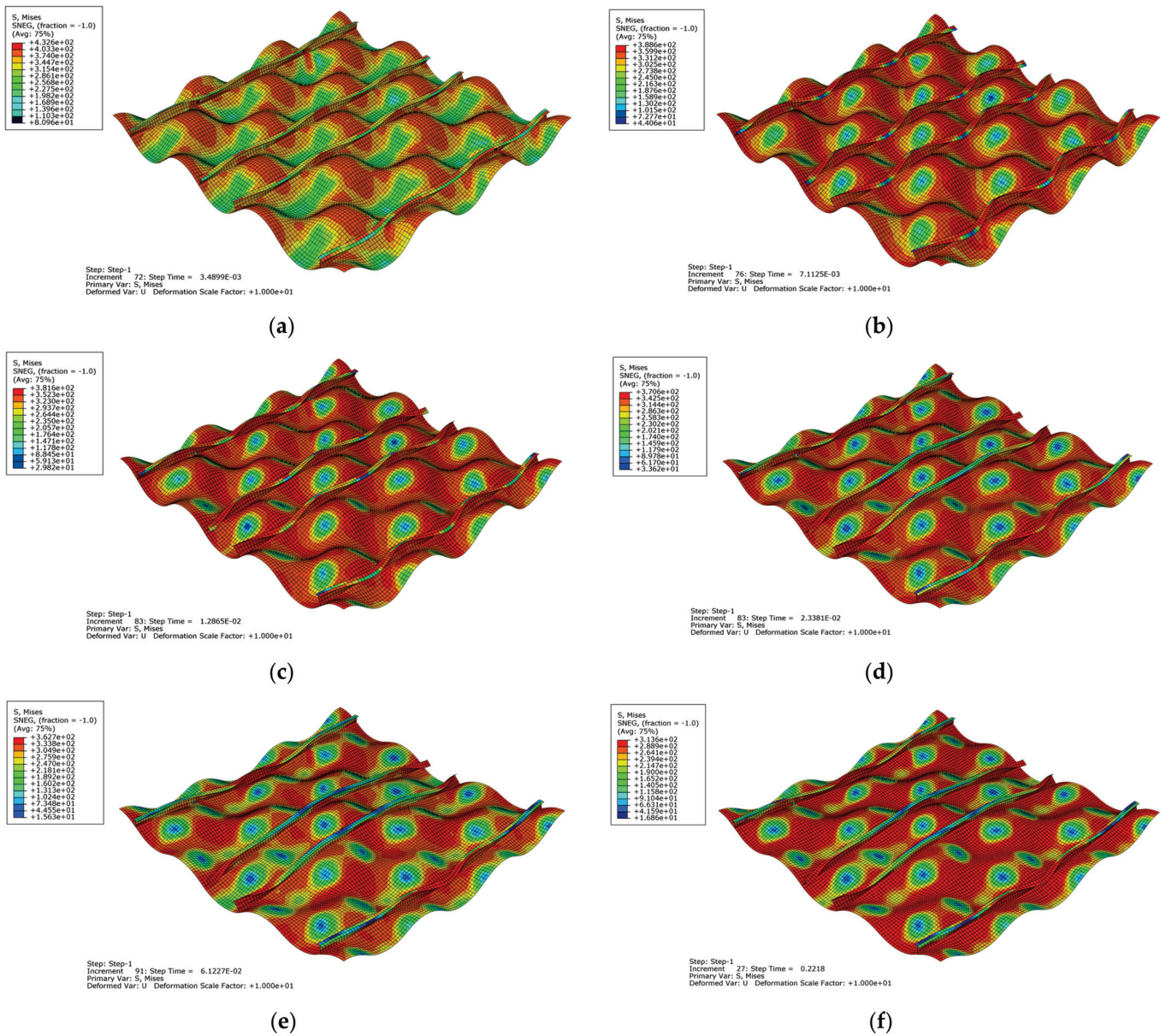


Figure 15. The diagram of deformation and stress distributions of Model 1 in the ultimate state. (a) $T_d/T_0 = 0.2$. (b) $T_d/T_0 = 0.5$. (c) $T_d/T_0 = 1.0$. (d) $T_d/T_0 = 2.0$. (e) $T_d/T_0 = 5.0$. (f) static ultimate state.

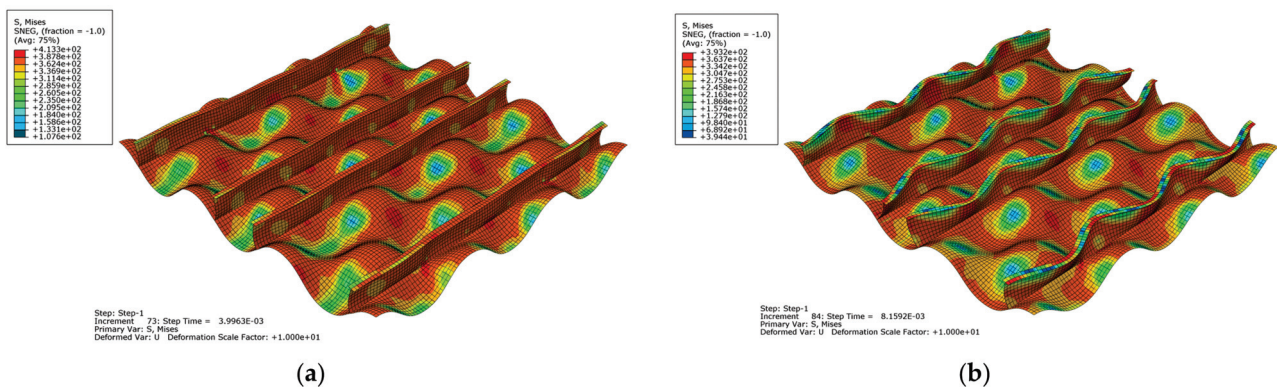


Figure 16. Cont.

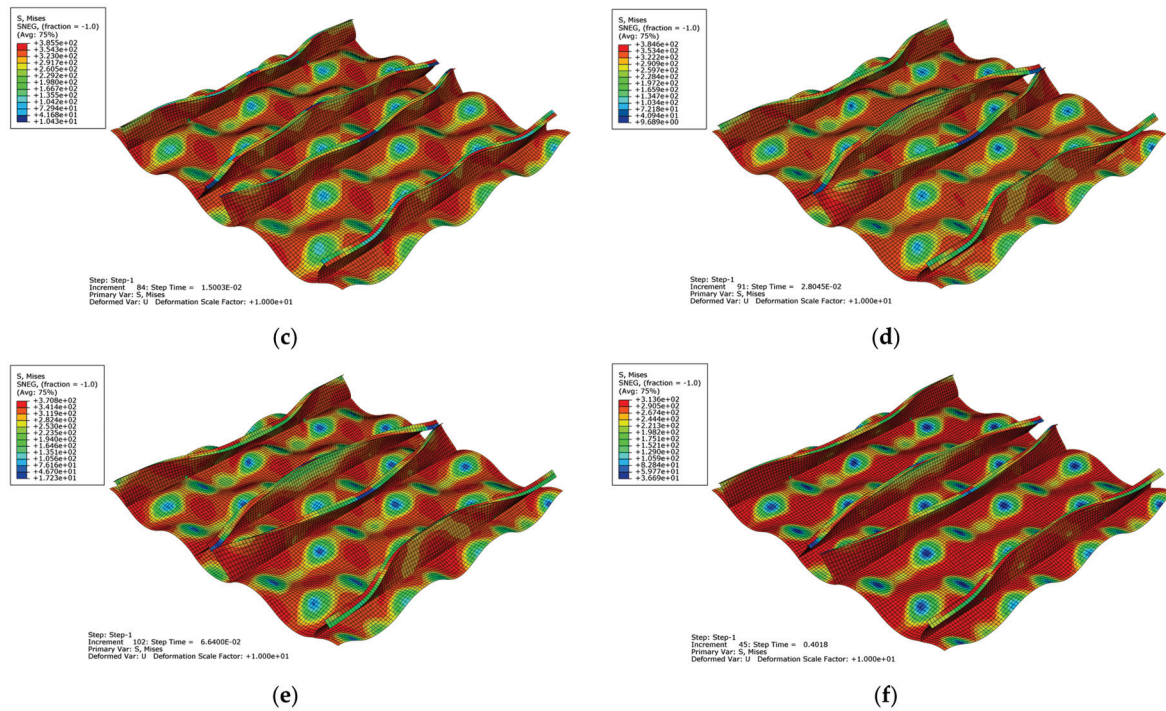


Figure 16. The diagram of deformation and stress distributions of Model 3 in the ultimate state. (a) $T_d/T_0 = 0.2$. (b) $T_d/T_0 = 0.5$. (c) $T_d/T_0 = 1.0$. (d) $T_d/T_0 = 2.0$. (e) $T_d/T_0 = 5.0$. (f) static ultimate state.

In summary, the impact duration can affect the dynamic failure mode of the stiffened plate. The dynamic failure mode of the stiffened plate for $T_d/T_0 = 5.0$ is the same as the static failure mode.

3.5. Effect of the Number of Impact Load Cycles on Dynamic Ultimate Load Capacity

Figure 17 shows the schematic diagram of the impact load cycles: one cycle, two cycles and three cycles. Using the two-step approach, the dynamic ultimate load capacity of Model 2 under different numbers of impact load cycles is calculated when $T_d = T_0$. Figure 18 shows the calculation results obtained by the two-step approach. As shown in Figure 18a, the greater the number of impact cycles, the larger the maximum axial displacement of the stiffened plate under an impact load of the same amplitude. This effect occurs because the greater the number of impacts, the larger the load impulse applied to the stiffened plate and the larger the maximum axial displacement. Based on the results shown in Figure 18b, the dynamic ultimate load capacity under one impact, two impacts and three impacts can be determined as $1.18\sigma_{u0}$, $1.16\sigma_{u0}$ and $1.12\sigma_{u0}$, respectively. Therefore, the dynamic ultimate load capacity decreases as the number of impact cycles increases.

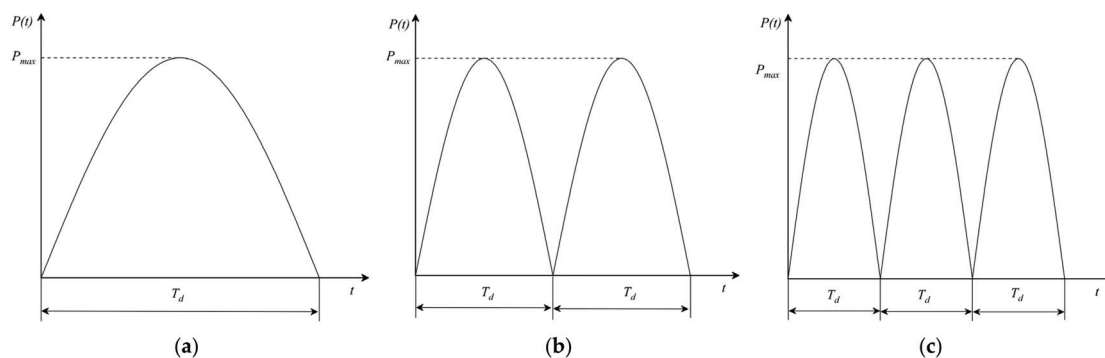


Figure 17. Schematic diagram of the impact load cycles. (a) one cycle. (b) two cycles. (c) three cycles.

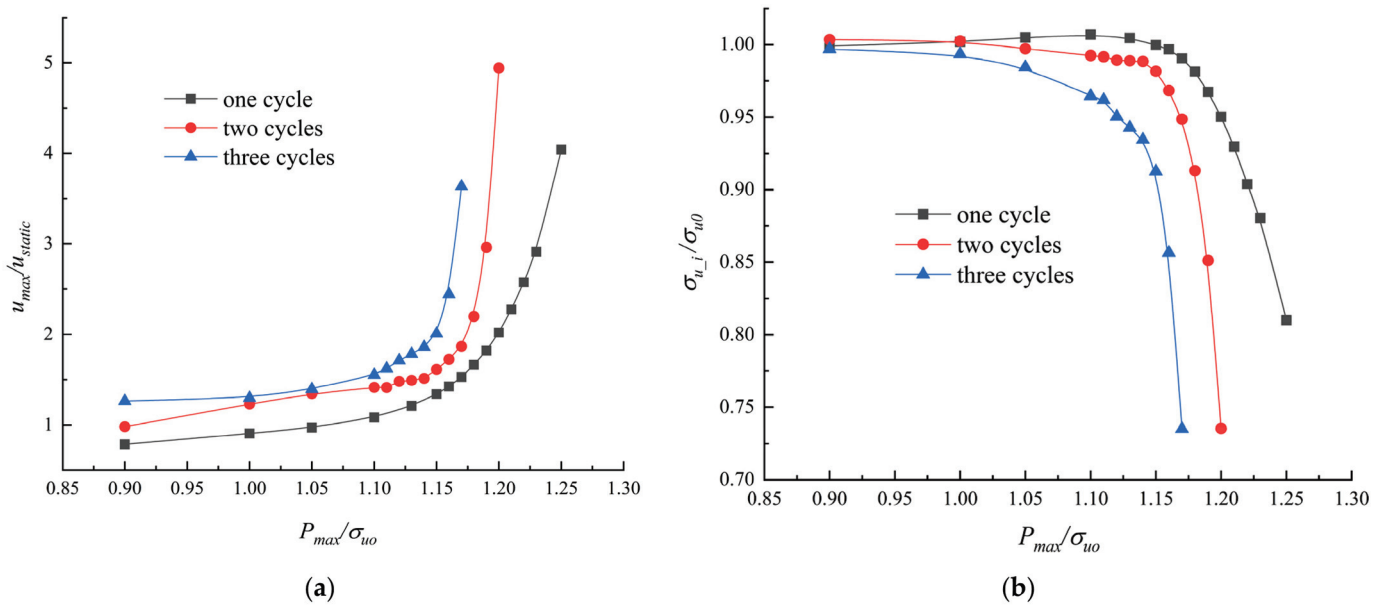


Figure 18. Calculation results of two-step approach under different numbers of impact load cycles. (a) The nondimensional maximum axial displacement versus nondimensional load amplitude. (b) The nondimensional static ultimate load capacity after impact versus nondimensional load amplitude.

3.6. Effect of the Impact Load Sequence on Dynamic Ultimate Load Capacity

Real ship structures may be subjected to continuous impact loads with different amplitudes. Therefore, it is meaningful to study the dynamic ultimate load capacity of ship structures under continuous impact. We assume that the stiffened plate is impacted three times continuously, each time with different amplitudes. As shown in Figure 19, there are a total of six different sequences of the impact load s . The duration of each impact is denoted as T_d . The three amplitudes of the impact load are $0.4P_{max}$, $0.6P_{max}$ and P_{max} , respectively. For ease of recording, the sequence of the impact load s is written as 1, 2, 3. For example, the sequence of the impact load amplitudes in Figure 19a is $0.4P_{max}$, $0.6P_{max}$, P_{max} , which can be denoted as “A-123”.

With Model 2 as the case study and static ultimate load capacity as the reference value, the dynamic ultimate load capacity is calculated by using the two-step approach when $T_d = T_0$. The calculation results are shown in Figure 20. As higher-order modes may be activated, as observed in the course of continuous impacts, the curve of the nondimensional static ultimate load capacity after impact versus nondimensional impact load impact becomes non-smooth. The dynamic ultimate load capacity is derived from the percentage reduction in the static ultimate load capacity after the impact. According to the above discussion, the static ultimate load capacity after impact cannot be reduced by more than 3%, so we have chosen 3% reduction the criterion for determining the dynamic load capacity. Thus, results for different impact load sequences can be obtained, as shown in Table 7. The 3% reduction is based on the nondimensional results from the five models of stiffened plates under five different impact times. Therefore, it is reasonable to take a 3% reduction as the criterion for determining the dynamic ultimate load capacity. When the last impact load is the one with the highest amplitude, i.e., when the load sequence is “A-123”, the dynamic ultimate load capacity of the stiffened plate is the smallest. Because the stiffened plate has been subjected to two impact loads with different amplitudes and then subjected to an impact load with a higher amplitude, the deformation and plastic region of the stiffened plate will be significantly larger than in other cases, reducing the dynamic ultimate load capacity. For comparison, when the load sequence is “A-321”, the highest-amplitude load occurs first. After that impact, the smaller loads are not able to expand the plastic region further. Additionally, plastic regions have already been generated

when the smaller loads occur, resulting in smaller deformations and a greater ultimate load capacity.

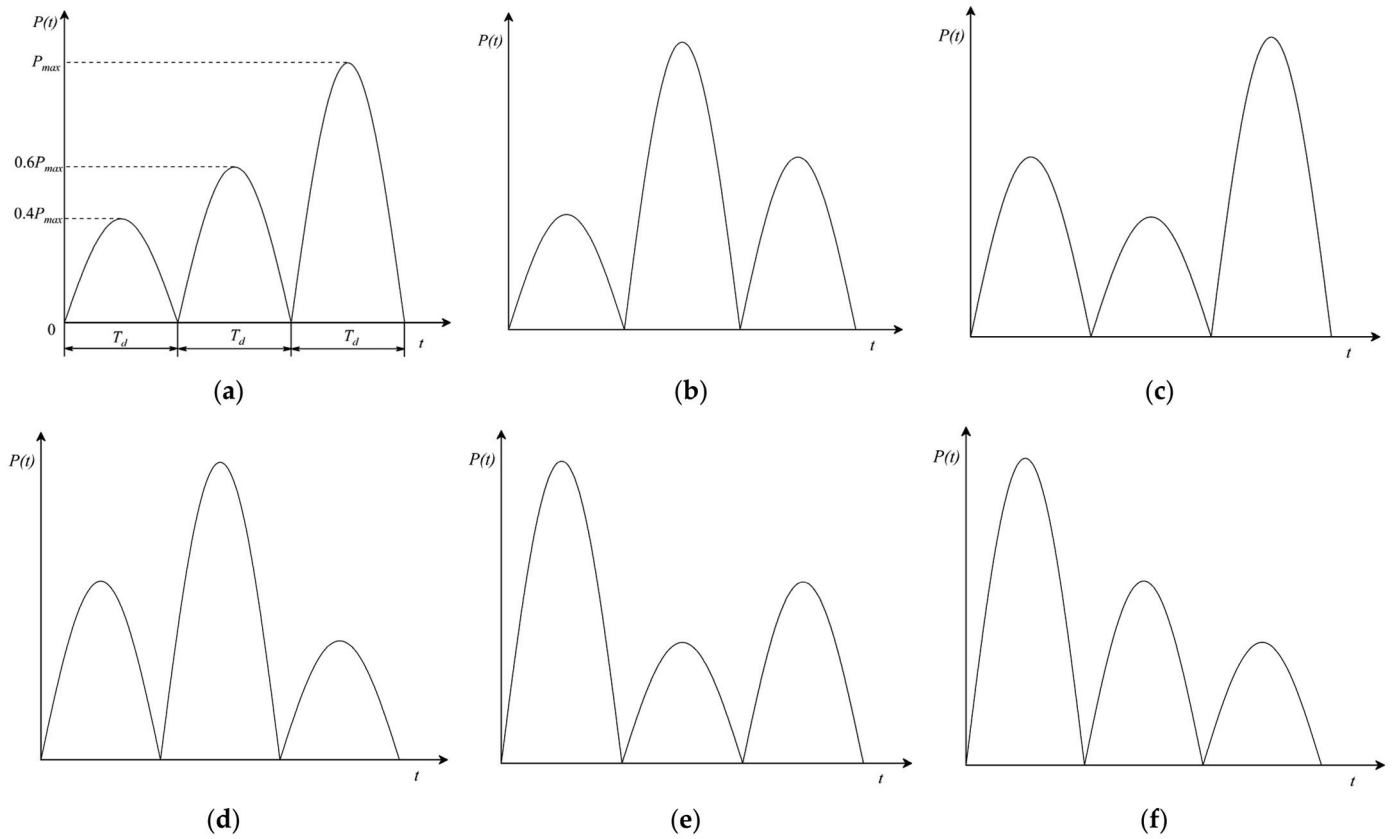


Figure 19. Shapes of continuous impact loads. (a) A-123. (b) A-132. (c) A-213. (d) A-231. (e) A-312. (f) A-321.

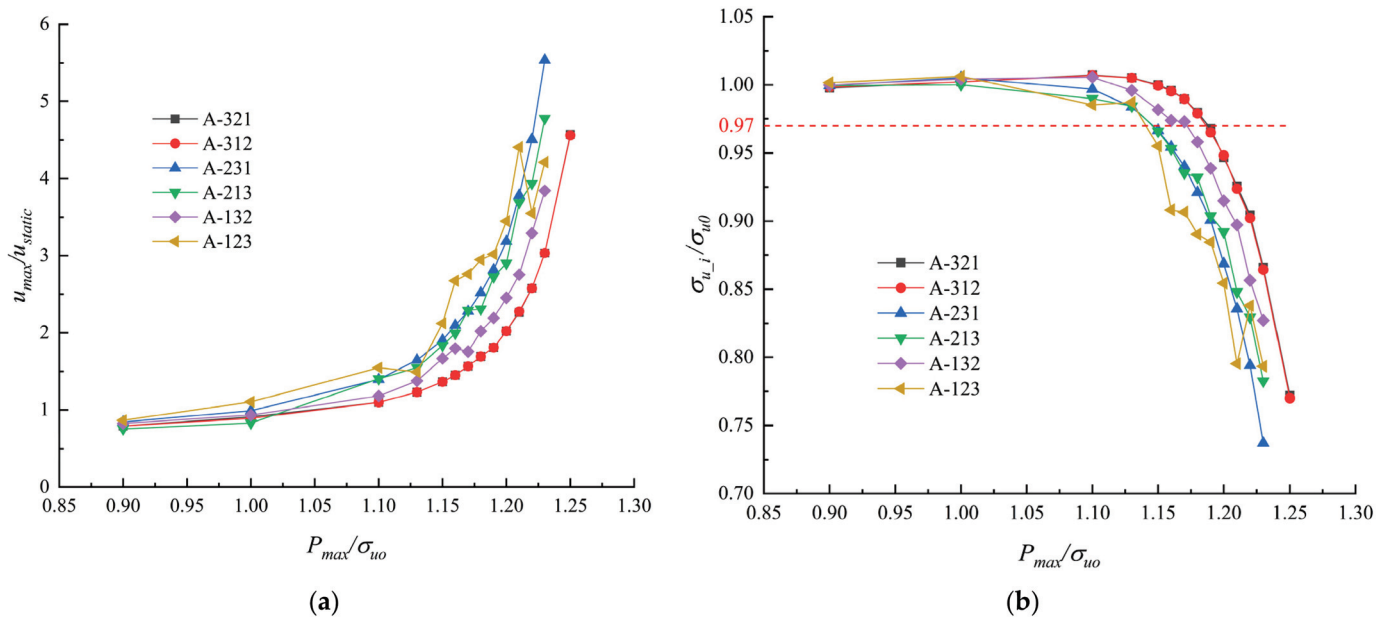


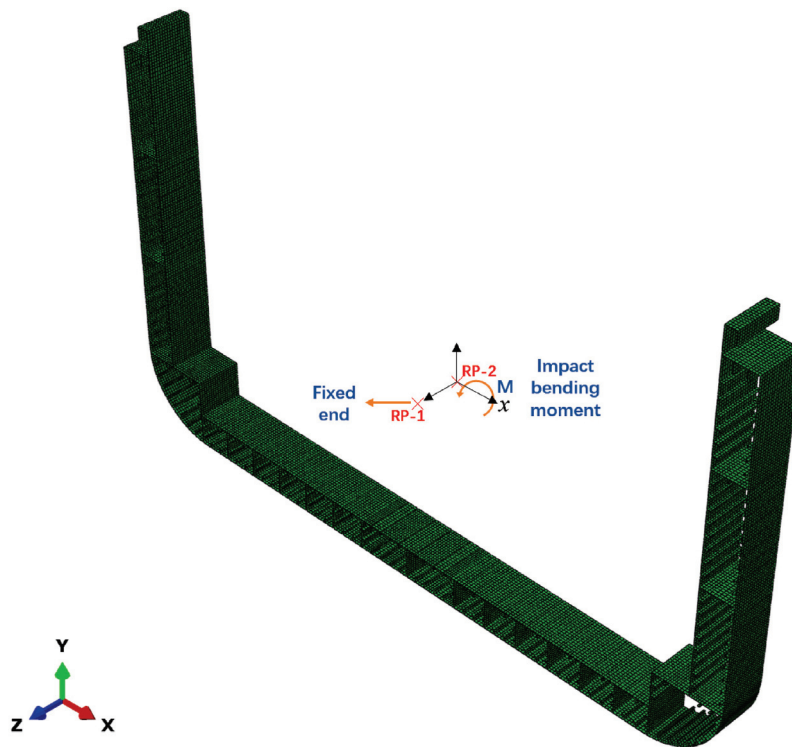
Figure 20. Calculation results of two-step approach under different load sequences. (a) The nondimensional maximum axial displacement versus nondimensional load amplitude. (b) The nondimensional static ultimate load capacity after impact versus nondimensional load amplitude.

Table 7. The dynamic ultimate load capacity of Model 1 under different load sequences.

Load Sequence	σ_{ud}/σ_{u0}
A-321	1.185
A-312	1.185
A-231	1.147
A-213	1.147
A-132	1.17
A-123	1.14

3.7. Application of the Two-Step Approach to a Hull Girder

In order to illustrate the applicability of the two-step approach to other ship structures, the two-step approach was used to calculate the dynamic ultimate load capacity of a hull girder. Figure 21 shows the schematic diagram of a single-frame model of a container-ship hull girder. The two reference points (RP-1 and RP-2) and all nodes at both ends of the model are coupled by coupling constraints. Both the boundary conditions and the impact bending moment are applied to the reference points. One end of the model is fixed, and the other end is subjected to impact bending moments. The shape of the impact bending moment is also half-sinusoidal, as shown in Figure 8. According to the natural vibration period of the single-frame model, $T_d = 0.05\text{s}$ is selected as the calculation case and the dynamic ultimate load capacity of the hull girder in the sagging state is calculated.

**Figure 21.** Schematic diagram of the single-frame model.

The procedure for calculating the dynamic ultimate load capacity of the hull girder by the two-step approach is similar to that used for the stiffened plates. First, the static ultimate load capacity M_{u0} and the static ultimate rotation angle UR_{static} of the hull girder are calculated. Thus, the nondimensional maximum dynamic response of the hull girder (the rotation angle of the cross section of the hull girder during impact.) and nondimensional amplitude of the impact bending moment can be defined as UR_{max}/UR_{static} and P_{max}/M_{u0} , respectively. The static ultimate load capacity of the hull girder after impact is denoted as M_{u_i} .

Figure 22 shows the results of calculating the dynamic ultimate load capacity of the hull girder by the two-step approach. From the trend shown by the curve in Figure 22b, it can be determined that the dynamic ultimate load capacity when $T_d = 0.05$ s is $1.18M_{u0}$ and that the static ultimate load capacity after the impact is reduced by 5%. In practice, ship designers can determine the percentage reduction in static ultimate load capacity after impact (e.g., 8% or 10%) according to their experience and needs. Therefore, the two-step approach can also be used to determine the dynamic ultimate load capacity of the hull girder.

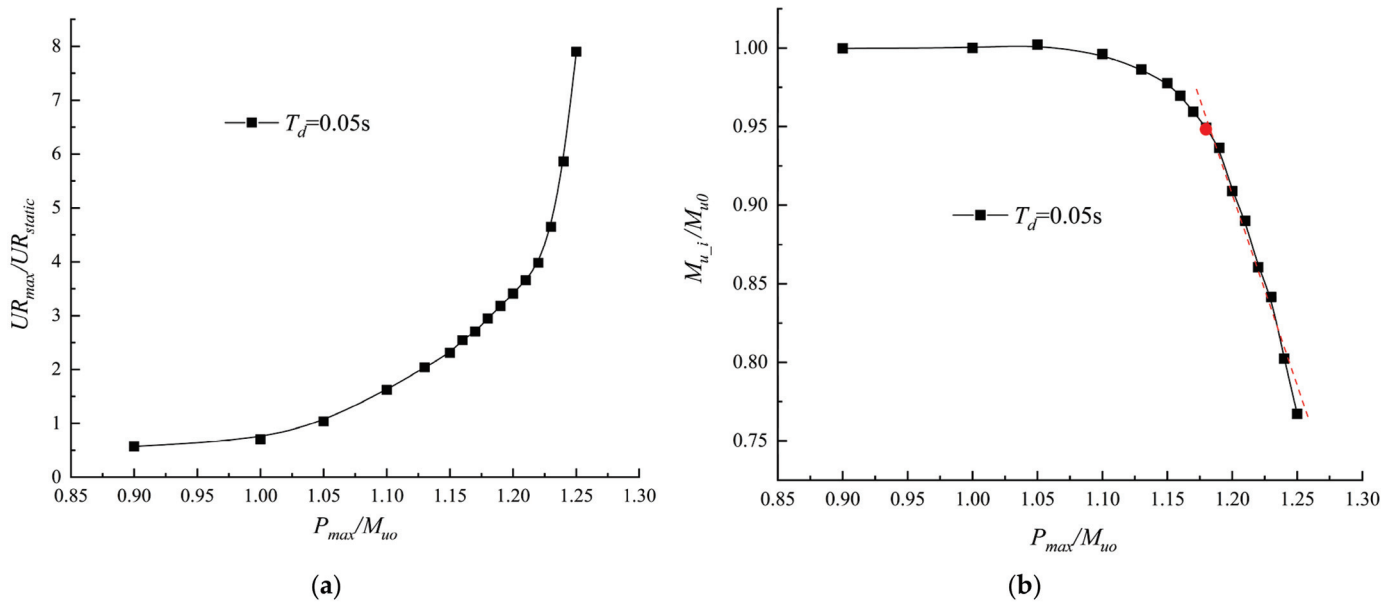


Figure 22. Calculation results from the two-step approach with $T_d = 0.05$ s. (a) The nondimensional maximum dynamic response versus nondimensional amplitude of the impact bending moment. (b) The nondimensional static ultimate load capacity after impact versus the nondimensional amplitude of the impact bending moment.

4. Conclusions

This study proposes a novel “two-step” approach for determining the dynamic ultimate load capacity. Taking a stiffened plate as a case study, the nonlinear FEM-based two-step approach is discussed in detail. Moreover, the applicability of the two-step approach to a hull girder is verified. The main conclusions of this study are summarized as follows:

(1) A criterion for the evaluation of the dynamic ultimate load capacity of ship structures is proposed in this paper based on the curve of nondimensional static ultimate load capacity after impact versus nondimensional impact load amplitude. These data can yield a reasonable determination of dynamic ultimate load capacity.

(2) The high-order deformation modes will be activated when $T_d/T_0 < 1$. Then, the deformation of the stiffened plate will increase and the dynamic ultimate load capacity of the stiffened plate will decrease.

(3) The static ultimate load capacity after the stiffened plate is subjected to an impact load whose amplitude is the dynamic ultimate load capacity is reduced by less than 3%.

(4) The failure mode of the stiffened plate will change as the impact duration changes. When $T_d/T_0 = 5.0$, the dynamic failure mode of the stiffened plate is the same as the static failure mode.

(5) The dynamic ultimate load capacity of the stiffened plate decreases as the number of impact cycles increases, while the impact duration remains constant.

(6) When the impact load with the highest amplitude is the last in the load sequence, the dynamic ultimate load capacity of the stiffened plate is the smallest; when the impact

load with the highest amplitude is first in the load sequence, the dynamic ultimate load capacity of the stiffened plate is the greatest.

Author Contributions: Conceptualization, J.C.; methodology, J.C.; software, W.Z.; validation, W.Z.; writing—review and editing, W.Z. and J.C.; supervision, D.W.; project administration, J.C. and D.W.; funding acquisition, J.C. and D.W. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by National Natural Science Foundation of China (NSFC, No. 52371328, No. 51809167 and No. 51979163 and U2241266) and the Fundamental Research Funds for the Central Universities.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors have no conflicts of interest to declare.

References

1. Smith, C.S. Influence of local compressive failure on ultimate longitudinal strength of a ship's hull. In Proceedings of the International Symposium on Practical Design in Shipbuilding (PRADS), Tokyo, Japan, 17–21 October 1977.
2. Tanaka, Y.; Ogawa, H.; Tatsumi, A.; Fujikubo, M. Analysis method of ultimate hull girder strength under combined loads. *Ships Offshore Struct.* **2015**, *10*, 587–598. [CrossRef]
3. Ueda, Y.; Rashed, S. An ultimate transverse strength analysis of ship structure. *J. Soc. Nav. Archit. Jpn.* **1974**, *136*, 309–324. [CrossRef]
4. Ueda, Y.; Rashed, S.M.H. The idealized structural unit method and its application to deep girder structures. *Comput. Struct.* **1984**, *18*, 277–293.
5. Lindemann, T.; Kaeding, P. Application of the idealized structural unit method for ultimate strength analyses of stiffened plate structures. *Ship Technol. Res.* **2017**, *64*, 15–29. [CrossRef]
6. Underwood, J.M.; Sobey, A.J.; Blake, J.I.R.; Sheno, R.A. Ultimate collapse strength assessment of damaged steel-plated structures. *Eng. Struct.* **2012**, *38*, 1–10. [CrossRef]
7. Zhang, S.; Khan, I. Buckling and ultimate capability of plates and stiffened panels in axial compression. *Mar. Struct.* **2009**, *22*, 791–808. [CrossRef]
8. Kim, D.K.; Lim, H.L.; Kim, M.S.; Hwang, O.J.; Park, K.S. An empirical formulation for predicting the ultimate strength of stiffened panels subjected to longitudinal compression. *Ocean Eng.* **2017**, *140*, 270–280. [CrossRef]
9. Xu, M.C.; Song, Z.J.; Zhang, B.W.; Pan, J. Empirical formula for predicting ultimate strength of stiffened panel of ship structure under combined longitudinal compression and lateral loads. *Ocean Eng.* **2018**, *162*, 161–175. [CrossRef]
10. Zhang, N.; Zong, Z. The effect of rigid-body motions on the whipping response of a ship hull subjected to an underwater bubble. *J. Fluids Struct.* **2011**, *27*, 1326–1336. [CrossRef]
11. Liu, W.; Suzuki, K.; Shibamura, K. Nonlinear dynamic response and structural evaluation of container ship in large freak waves. *J. Offshore Mech. Arct. Eng.* **2015**, *137*, 011601. [CrossRef]
12. Sumi, Y.; Fujikubo, M.; Fujita, H.; Kawagoe, Y.; Kidogawa, M.; Kobayashi, K.; Nakano, T.; Iwano, J.; Takahira, T.; Tamura, K.; et al. *Final Report of Committee on Large Container Ship Safety*; Committee on Large Container Ship Safety: Tokyo, Japan, 2015.
13. Yamada, Y. Dynamic collapse mechanism of global hull girder of container ships subjected to hogging moment. *J. Offshore Mech. Arct. Eng.* **2019**, *141*, 051605. [CrossRef]
14. Jagite, G.; Bigot, F.; Malenica, S.; Derbanne, Q.; Sourne, H.L.; Cartraud, P. Dynamic ultimate strength of a ultra-large container ship subjected to realistic loading scenarios. *Mar. Struct.* **2022**, *84*, 103197. [CrossRef]
15. Yang, B.; Soares, C.G.; Wang, D.Y. An empirical formulation for predicting the dynamic ultimate strength of rectangular plates under in-plane compressive loading. *Int. J. Mech. Sci.* **2018**, *141*, 213–222. [CrossRef]
16. Yang, B.; Wang, D.Y. Dynamic buckling of stiffened plates with elastically restrained edges under in-plane impact loading. *Thin-Walled Struct.* **2016**, *107*, 427–442. [CrossRef]
17. Paik, J.K.; Thayamballi, A.K. An experimental investigation on the dynamic ultimate compressive strength of ship plating. *Int. J. Impact Eng.* **2003**, *28*, 803–811. [CrossRef]
18. Liu, B.; Villavicencio, R.; Soares, C.G. Experimental and numerical plastic response and failure of laterally impacted rectangular plates. *J. Offshore Mech. Arct. Eng.* **2013**, *135*, 041602. [CrossRef]
19. Jagite, G.; Bigot, F.; Derbanne, Q.; Malenica, S.; Sourne, H.L.; Cartraud, P. A parametric study on the dynamic ultimate strength of a stiffened panel subjected to wave-and whipping-induced stresses. *Ships Offshore Struct.* **2021**, *16*, 1025–1039. [CrossRef]
20. Ari-Gur, J.; Simonetta, S.R. Dynamic pulse buckling of rectangular composite plates. *Compos. B Eng.* **1997**, *28*, 301–308. [CrossRef]

21. Xiong, Y.; Li, C.; Cai, S.; Wang, D. The dynamic ultimate strength of stiffened panels under axial impact loading. *Ships Offshore Struct.* **2023**, *18*, 707–720. [CrossRef]
22. Yang, S.H.; Lee, Y.J.; Chien, H.L.; Chou, C.M.; Tseng, K.C. Dynamic Buckling Strength Assessment of Containership's Bow Structures Subjected to Bow Flare Slamming. In *ASME International Mechanical Engineering Congress and Exposition*; American Society of Mechanical Engineers: New York, NY, USA, 2012; pp. 727–733.
23. Yang, B.; Soares, C.G.; Wang, D.Y. Dynamic ultimate compressive strength of simply supported rectangular plates under impact loading. *Mar. Struct.* **2019**, *66*, 258–271. [CrossRef]
24. Jagite, G.; Bigot, F.; Derbanne, Q.; Malenica, Š.; le Sourne, H.; de Lauzon, J.; Cartraud, P. Numerical investigation on dynamic ultimate strength of stiffened panels considering real loading scenarios. *Ships Offshore Struct.* **2019**, *14*, 374–386. [CrossRef]
25. Paik, J.K.; Seo, J.K. Nonlinear Finite Element Method Models for Ultimate Strength Analysis of Steel Stiffened-Plate Structures Under Combined Biaxial Compression and Lateral Pressure Actions—Part II: Stiffened Panels. *Thin-Walled Struct.* **2009**, *47*, 998–1007. [CrossRef]
26. Paik, J.K.; Kim, B.J.; Seo, J.K. Methods for Ultimate Limit State Assessment of Ships and Ship-Shaped Offshore Structures: Part II—Stiffened Panels. *Ocean Eng.* **2008**, *35*, 271–280. [CrossRef]
27. Cowper, G.R.; Symonds, P.S. *Strain-Hardening and Strain-Rate Effects in the Impact Loading of Cantilever Beams*; Technical Report 28; Division of Applied Mathematics, Brown University: Providence, RI, USA, 1957.
28. Paik, J.K. *Ultimate Limit State Analysis and Design of Plated Structures*, 2nd ed.; John Wiley & Sons: Chichester, UK, 2018.
29. Paik, J.K.; Kim, D.K.; Kim, M.S. Ultimate strength performance of Suezmax tanker structures: Pre-CSR versus CSR designs. In *Analysis and Design of Marine Structures*, 1st ed.; Soares, C.G., Das, P.K., Eds.; Taylor & Francis Group: London, UK, 2009; pp. 181–190.
30. Paik, J.K.; Sohn, J.M. Effects of welding residual stresses on high tensile steel plate ultimate strength: Nonlinear finite element method investigations. *J. Offshore Mech. Arct. Eng.* **2012**, *134*, 021401. [CrossRef]
31. Chen, B.Q.; Soares, C.G. Effects of plate configurations on the weld induced deformations and strength of fillet-welded plates. *Mar. Struct.* **2016**, *50*, 243–259. [CrossRef]
32. Paik, J.K.; Amlashi, H.; Boon, B.; Branner, K.; Caridis, P.; Das, P.; Fujikubo, M.; Huang, C.-H.; Josefson, L.; Kaeding, P.; et al. Committee III.1 Ultimate strength. In *Proceedings of the 18th International Ship and Offshore Structures Congress (ISSC)*, Rostock, Germany, 10–13 September 2012; Volume 1, pp. 285–363.
33. Smith, C.S.; Davidson, P.C.; Chapman, J.C. Strength and stiffness of ships' plating under in-plane compression and tension. *R. Inst. Nav. Archit. Trans.* **1987**, *130*, 277–296.
34. Paik, J.K.; Kim, B.J. Ultimate strength formulations for stiffened panels under combined axial load, in-plane bending and lateral pressure: A benchmark study. *Thin-Walled Struct.* **2002**, *40*, 45–83. [CrossRef]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Experimental and Numerical Studies on the Ultimate Bending Strength of Welded Plated Grillage with Combined Openings

Chen Chen *, Hong Zhou *, Zhengda Lv and Ziqiu Li

School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology,
Zhenjiang 212100, China; 15142839625@163.com (Z.L.); 221110101119@stu.just.edu.cn (Z.L.)

* Correspondence: 201010006@stu.just.edu.cn (C.C.); 199600001373@just.edu.cn (H.Z.)

Abstract: Plated grillage with combined openings was susceptible to complex failure behaviors as the main load-bearing structure of the superstructure on passenger ships subjected to deck loads. Additionally, the deformation and stresses generated during the welding of the plated grillage complicated the prediction of its failure behavior. In this case, a new partitioned inherent strain method and nonlinear finite element method were used to simulate the welding and loading process, and experiments were designed and carried out to make comparisons, unveiling the influence regulations between the failure behavior of the structure and the loading condition, the initial welding state. This research on the failure mode analysis of plated grillages could provide references for the optimization of the structural form of plated grillages and the cargo loading scheme on the deck of a real ship.

Keywords: combined opening plated grillage; initial welding state; ultimate bending strength; failure behavior

1. Introduction

Large passenger ships are usually composed of multideck plated grillage structures constructed from welded and assembled plates and sections. Differing from conventional ships, to meet the requirements of dense pipeline and cable arrangement as well as accommodation layout, the plated grillage is structured with a large span, high stiffener web height, and dense openings [1,2]. In addition, to reduce the weight, the plated grillage in the accommodation area is made of numerous thin plates, causing significant deformation during the welding process [3]. Structural specializations and welding deformations make the deformation behavior of such plated grillage structures complicated when subjected to vertical loads. Therefore, it is necessary to investigate the ultimate load-bearing capacity and deformation characteristics of welded combined opening plated grillage under deck load, taking into account the welding deformation and stress.

To investigate behaviors of the opening plate and girders, related scholars have made numerous explorations. For opening plates, Brown [4] first solved for the stress distribution around the perimeter of the opening using the conjugate load analysis method and investigated the effect of different opening shapes [5,6]. Later, with the evolution of computer hardware and the iteration of algorithms, finite element calculation methods were widely used. Shanmugam, Paik, and other scholars investigated the effects of opening shape, size, location, and loading method on the deformation behavior and ultimate load-bearing capacity with the aid of FEM, and summarized the designing equations [7–12]. As for the experiment, Saad-Eldden systematically investigated the effects of opening shape, size, reinforcement measure, cracking, and material corrosion on structural load-bearing capacity, and found that when the opening size is constant, small-size dense openings could alleviate the localized failure phenomenon occurring in large-size sparse openings [13–18].

For the structural and more complex combined opening girders, scholars mostly used finite element simulation and experimental methods to explore deformation regulations

and load-bearing characteristics. Sun et al. [19–21] used the FEM to investigate the failure modes of opening girders made of concrete under torsion, bending, and shear loads, and gave the corresponding reinforcement measures. Wang et al. [22] investigated the load-bearing capacity of steel hexagonal opening girders under vertical shear loads, revealing the deformation characteristics. As for the experiments, Zirakian et al. [23] carried out a load-bearing capacity experiment on hexagonal opening girders, compared the results with the Trahair formula [24], and gave suggestions for improvements for the formula generalization. Zhao et al. [2,25] designed an experimental device to carry out the ultimate load-bearing capacity of a ship's opening girder structure under the combination of axial and vertical loads, and suggested adding a guard plate around the opening to improve the capacity of the opening region.

For marine combined opening plated grillage, owing to the structural complexity, most investigations were still based on the finite element method. Li et al. [26] took the opening plated grillage on a ro-ro ship as a research object, and used the FEM to explore the influence regulations of the openings on the overall and local strength of the plated grillage, and provided suggestions for the optimization of the structure. Gan et al. [27] analyzed the effects of the initial defects and framing specifications on the ultimate bearing capacity of the opening plated grillage under longitudinal compression loads, combining the classical reinforced plate theory and the nonlinear finite element method. As for the experiments, Liu et al. [28,29] conducted the design of the loading device to investigate the failure behavior of the passenger ship opening plated grillage under axial compression loads, revealing the influence regulations of the initial defects, opening sizes, and some other parameters.

In the above-mentioned literature, there are relatively few experimental investigations on the ultimate load-bearing capacity of high stiffener web plated grillage with combined openings. For loading conditions, the action of the axial compression load was mainly considered, and little consideration was given to the deck bending load. In addition, regardless of the research on girders or plated grillages, most of the considerations for the initial welding state in the simulation were still based on empirical formulas, and the experiments were still dominated by corrective elimination. However, the passenger ship's plated grillage uses a large number of thin plates, which are prone to distinct deformation at the welding joint. Due to the difference in welding parameters, the deformation and residual stress values also differ greatly [30]. The use of empirical formulas to impose a homogenized initial defect does not reflect the detailed manufacturing process and is subjected to large errors. While using the inherent strain method, the deformation of the plated grillage can be accurately and efficiently predicted considering the welding parameters. Zhou et al. [31–34] used this approach to perform a deformation prediction of offshore platform blocks and gave process optimization measures. Therefore, the welding deformation and stress of the plated grillage can be predicted using the inherent strain method and used as the initial state for the simulation of the failure behavior to improve the accuracy of the numerical simulation.

In this study, the load-bearing capacity and failure behavior of combined opening plated grillage were investigated using numerical simulations combined with static experiments. An inhomogeneous inherent strain method was used to predict the initial stresses and deformations of the plated grillage, as an initial state to study the effect of the initial welding state on the failure behavior of the structure. Finally, the ultimate load-bearing capacity as well as the failure behavior were investigated via experimental methods combined with simulated results.

2. Specimens

To satisfy the demand for space arrangement in the theater area of the large passenger ship, a large-span combined opening plated grillage is used in conjunction with a pillar to achieve structural support. The plated grillage between the two longitudinal girders in this region was selected as the object to investigate its failure behavior under deck

vertical loads. The plated grillage was assembled and welded from longitudinal girders and stringers, transverse, and plates. The location of the plated grillage on the actual ship was shown in Figure 1. Referring to the existing research findings [28,29] and the site equipment conditions, the geometric and plate thickness scaling ratios were determined to be 1:4 and 1:2, respectively. The dimensions of the plated grillage and the parts were shown in Figure 2.

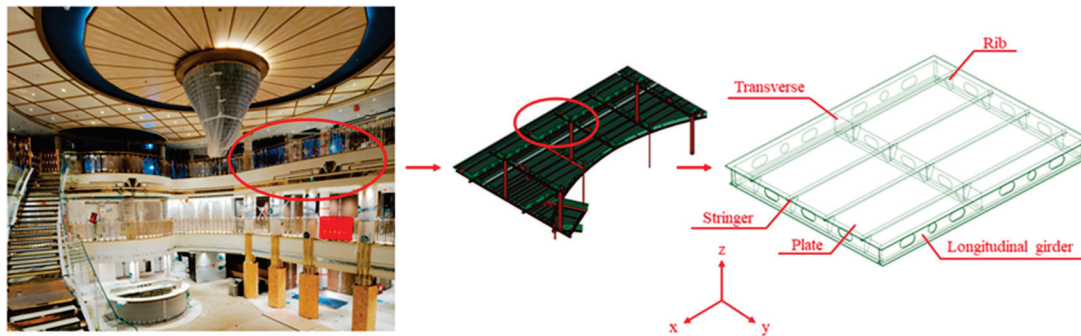


Figure 1. Schematic diagram of the plated grillage structure.

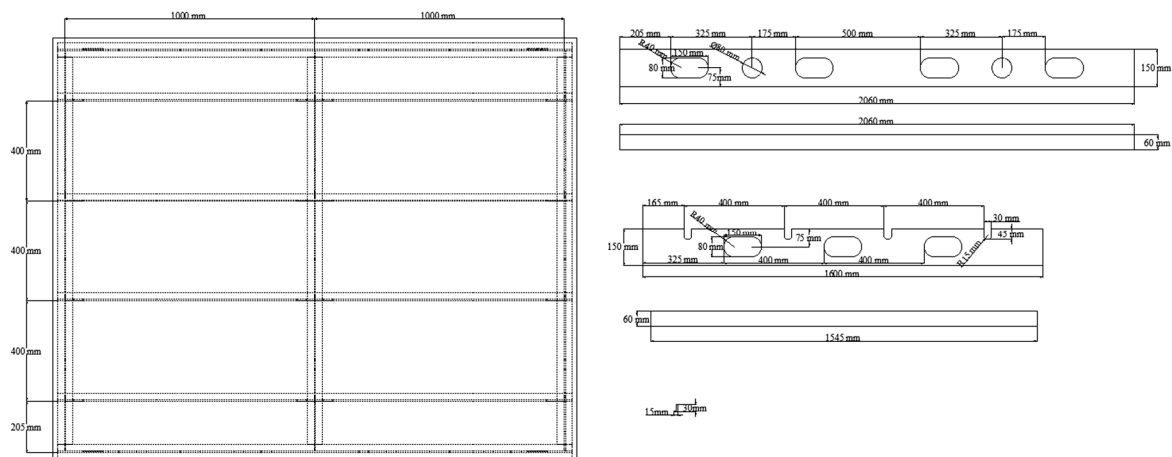


Figure 2. Diagram of the plated grillage dimensions (mm).

3. Simulation and Measurement of Initial Welding State

To better investigate the effect of the initial welding state in the manufacturing on the failure behavior of the structure, an inhomogeneous inherent strain method was used in this section to simulate the deformation and stress of the plated grillage concerning the actual manufacturing process. The deformation values of the measured points on the deck surface obtained from the simulation were compared with the measured values to verify the accuracy of the numerical simulation.

3.1. The Actual Manufacturing Technique

The plated grillage was assembled and welded from an AH36 mild steel of 5 mm. For the welding process, the structure was welded using CO₂ gas-shielded welding. The equipment used was an inverter-type semi-automatic CO₂/MAG arc welding machine with a gas flow rate of 15–20 L/min. The wire used was YC-YJ502 (Q) flux-cored wire with a wire diameter of 1.2 mm. Due to the thin plate thickness, no bevels were needed before welding. Welding joints were formed in one pass, with no need for multiple welds. The weld was first completed on one side of the T connection, then the torch was flipped over to the other side of the welding joint, followed by flipping the torch to weld the other side. The current, voltage, and welding speed for the two welds were shown in Table 1. For the welding sequence, the welding of the stringers with the plate was performed first, followed

by the welding of the longitudinal girders with the plate, and finally the welding of the transverse with the plate. The welding sequence was shown in Figure 3.

Table 1. Table of welding parameters.

No.	Current (A)	Voltage (V)	Velocity (mm/s)
1	142–155	22–25	6.2
2	138–147	20–22	6.1

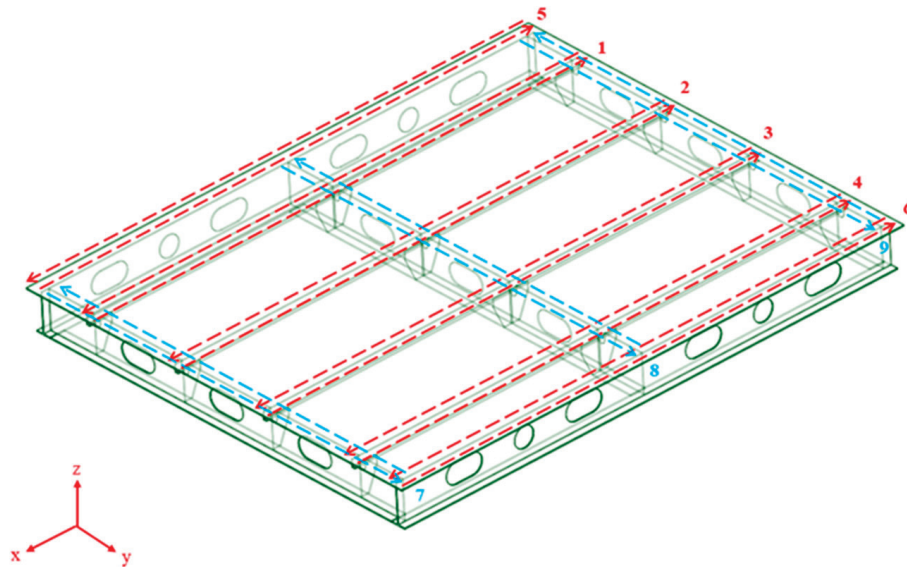


Figure 3. Diagram of the welding sequence.

3.2. Simulation of the Deformation and Stress of the Plated Grillage

Referring to actual welding data, the inherent deformation of the T connection was evaluated using the thermo-elastic-plastic finite element method. On this basis, the welding joint of the plated grillage was divided into parts, and the inherent deformation was applied to obtain the overall deformation and stress distribution.

3.2.1. Calculation Method

Welding of the steel is a process of melting and re-solidification at the atomic level [33]. During the welding process, the presence of a large temperature gradient between the weld region and surrounding base material leads to an uneven distribution of residual plastic strain, which leads to the bending moment and out-of-plane welding deformation of the structure.

Currently, the two main finite element simulation methods for the welding process are the thermo-elastic-plastic finite element method and the inherent strain method. In this case, the thermo-elastic-plastic element method is first used to calculate the thermal loads applied to the mesh nodes from the heat source employing heat transfer and heat radiation. Subsequently, the mesh nodes on the structure are subjected to thermal loads generating displacements so that the stresses and deformations of the structure can be evaluated from the material's intrinsic relationship. This method can evaluate the stress and deformation of each mesh node throughout the welding process, but it is computationally expensive and takes a long time to compute for large structures [35].

To realize the rapid prediction of deformation and stress in large structures, Ueda et al. [36] conducted a series of experiments and theoretical derivations, and proposed an analysis method based on inherent strain, i.e., it is considered that the inherent strain is the fundamental cause of welding deformation and residual stress. Inherent strains $\epsilon^{inherent}$

include thermal stress $\varepsilon^{thermal}$, plastic strains $\varepsilon^{plastic}$, creep strains ε^{creep} , and transformation strains $\varepsilon^{transformation}$, as shown in Equation (1).

$$\varepsilon^{inherent} = \varepsilon^{thermal} + \varepsilon^{plastic} + \varepsilon^{creep} + \varepsilon^{transformation} \quad (1)$$

Since thermal strains disappear when the structure is cooled, and creep and transformation strains occur in small quantities compared to plastic strains, it is approximated in the calculations that the inherent strains are equal to the plastic strains. And due to the inhomogeneity of the plastic strain distribution, some simplifications are made in engineering applications. Due to the long length of the welding joint, the end effect is usually ignored, and the welding joint region is sectioned into separate areas to apply the inherent deformation to simulate the stresses and deformations generated in the whole structure [37]. During the finite element simulation, the strain can be converted to a thermal expansion coefficient according to Equation (2). The calculation of structural deformation and stress can be achieved by applying a unit temperature difference to the region of the welding joint where the inherent strain needs to be applied.

$$\varepsilon_T = \alpha \cdot \Delta T \quad (2)$$

where, ε_T is the inherent strain value, α is the thermal expansion coefficient, and ΔT is the unit temperature difference.

3.2.2. Calculation of the Inherent Deformation Data of Joints

The solid element model of the T joint was established, with a length and width of 400 mm, and a height of 250 mm. The height of the weld foot was 3 mm. The melt pool region was mesh-encrypted, and the number of elements and nodes were 24,950 and 34,218, respectively. The boundary conditions were set, as shown in Figure 4.

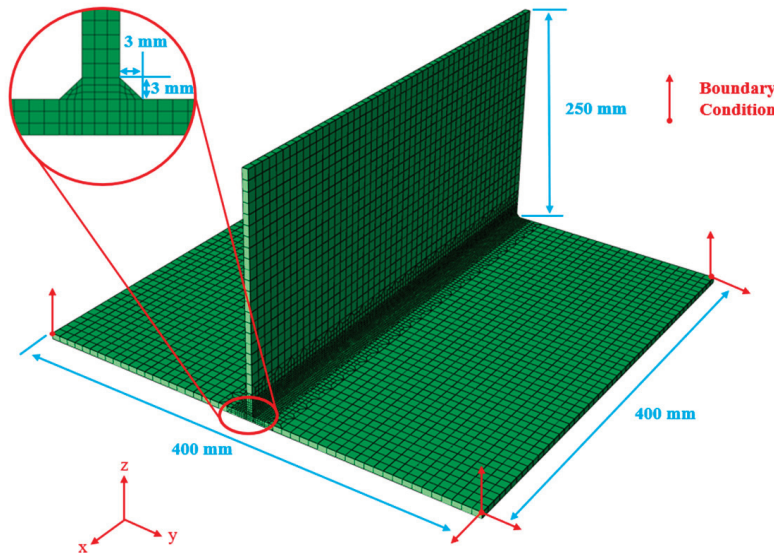


Figure 4. Diagram of solid element model of the T joint.

Based on the completion of the above setup and recorded actual welding parameters (as Table 1), the thermo-elastic-plastic finite element calculation of the T-joint was performed. In this section, the sequential coupling calculation method was adopted, i.e., the temperature field calculation was carried out first, and the stress and deformation of the joint were obtained by applying the temperature load to the mesh nodes after the calculation was completed. The material parameters of AH36 were shown in Figure 5.

For the computed temperature, the maximum temperature of the joint was 2075 °C. As was shown in Figure 6, the region where the temperature was higher than the melting point of the steel was essentially the same as the geometry of the melt pool. Then, the

temperature load was applied to the structure and the computed deformation as well as plastic strain distribution was shown in Figures 7 and 8. In Figures 7 and 8, NT11 is the nodal temperature ($^{\circ}\text{C}$), U is the deformation (mm), and PE11, PE22, and PE33 are the plastic strains in the x, y, and z directions, respectively.

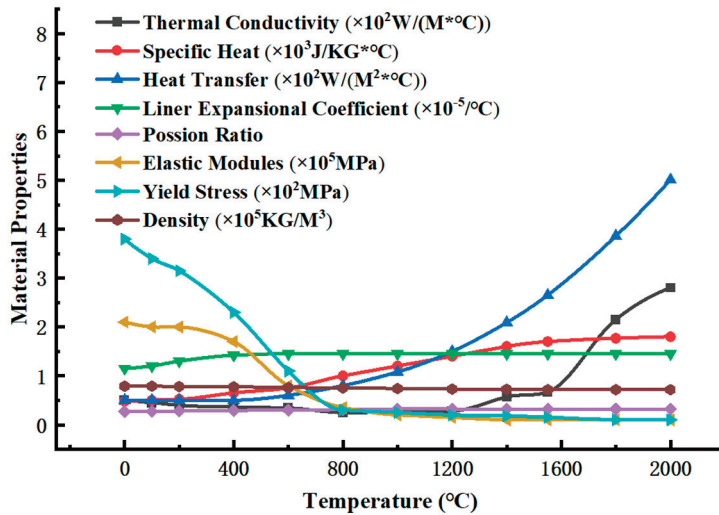


Figure 5. Diagram of the material parameters of AH36.

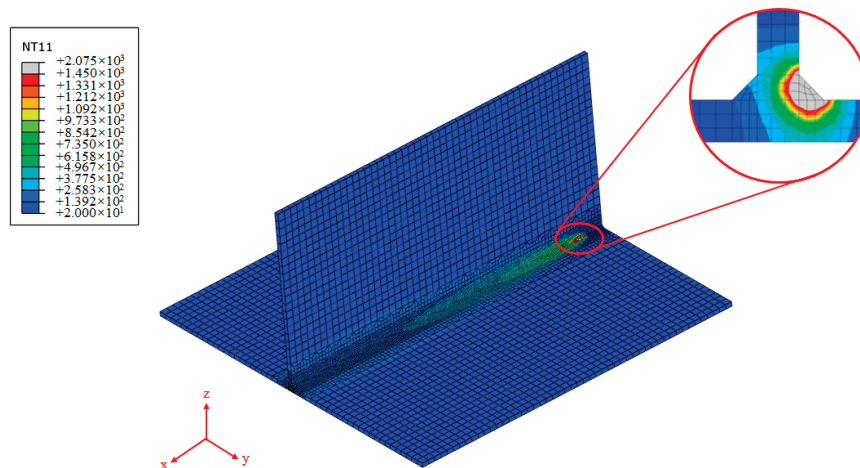


Figure 6. Diagram of the computed temperature distribution.

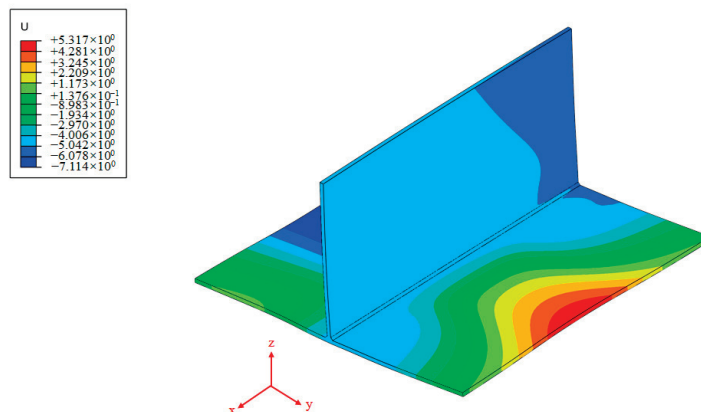


Figure 7. Diagram of the computed deformation.

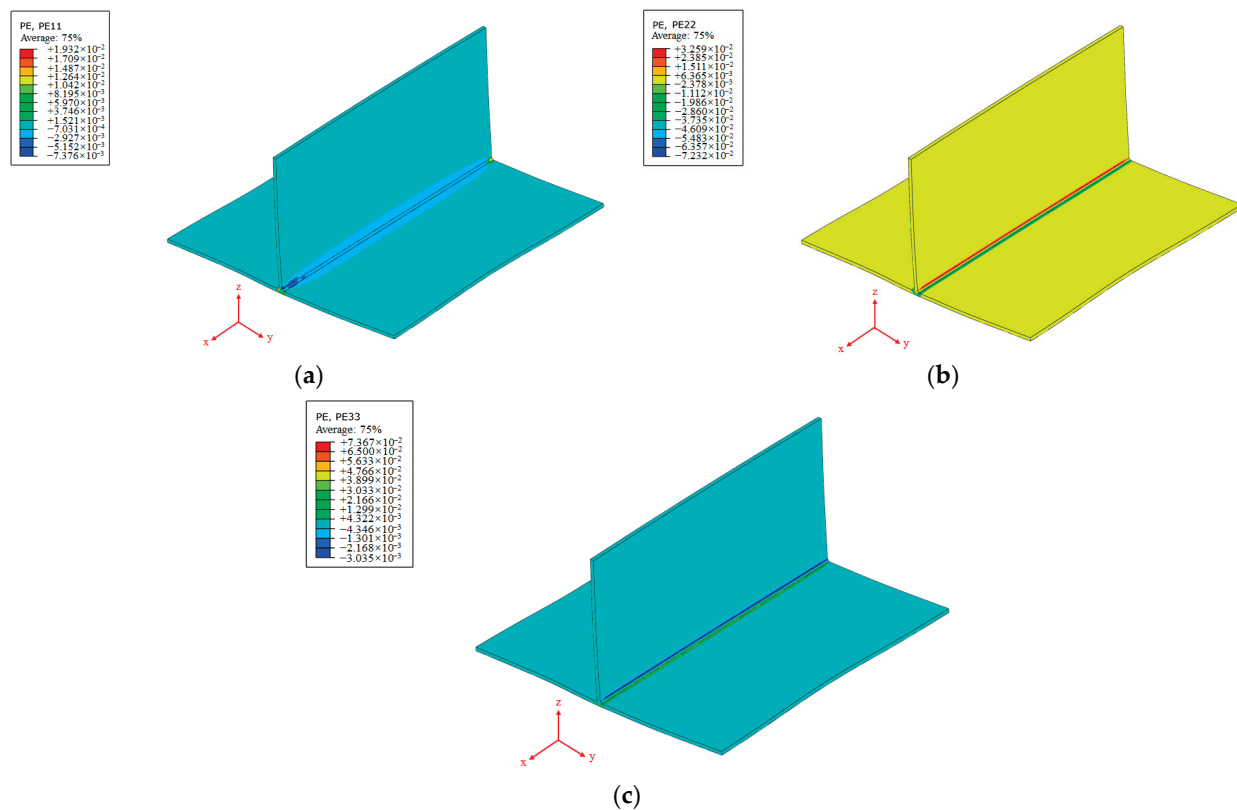


Figure 8. Diagram of the computed plastic strain distribution. (a) Longitudinal plastic strain, (b) transverse plastic strain, and (c) vertical plastic strain.

As was shown in Figure 8, the plastic strain values around the welding joint were higher. In the following section, the plastic strain in this region was extracted and converted to thermal expansion coefficients referring to Equation (2) to be assigned to the welding joint regions of the plated grillage for the calculation of its deformation and stress distribution.

3.2.3. Welding Deformation and Stress Computation of Plated Grillage

On the basis of the computation to obtain the joint inherent strain data in Section 3.2.2, the application of the inherent strain on the plated grillage was carried out. After analyzing the results of the T joint computation, it was found that the plastic strain distribution in the molten pool region was distinctly inhomogeneous in the Z-direction. Particularly for plastic strains in the Y- and Z-directions, there was a distinct difference between tensile and compressive strains (as shown in Figure 9).

Due to the inhomogeneous distribution characteristics of the plastic strain at the weld obtained from the joint calculations along the Z-direction, the plated grillage was subjected to the operation of partitioning and applying different inherent deformations to obtain a higher accuracy of calculation. The partitioning approach for the welding joints between the plate and longitudinal girder, stringer, and transverse was shown in Figure 10. The thermal expansion coefficients of the material corresponding to the unit temperature difference obtained referring to Equation (2) were shown in Table 2.

Table 2. Table of the thermal expansion coefficients.

Direction	Thermal Expansion Coefficients	
	Region-A	Region-B
Longitudinal	−0.00233	−0.00248
Transverse	−0.010	0.016
Vertical	0.0126	−0.0254

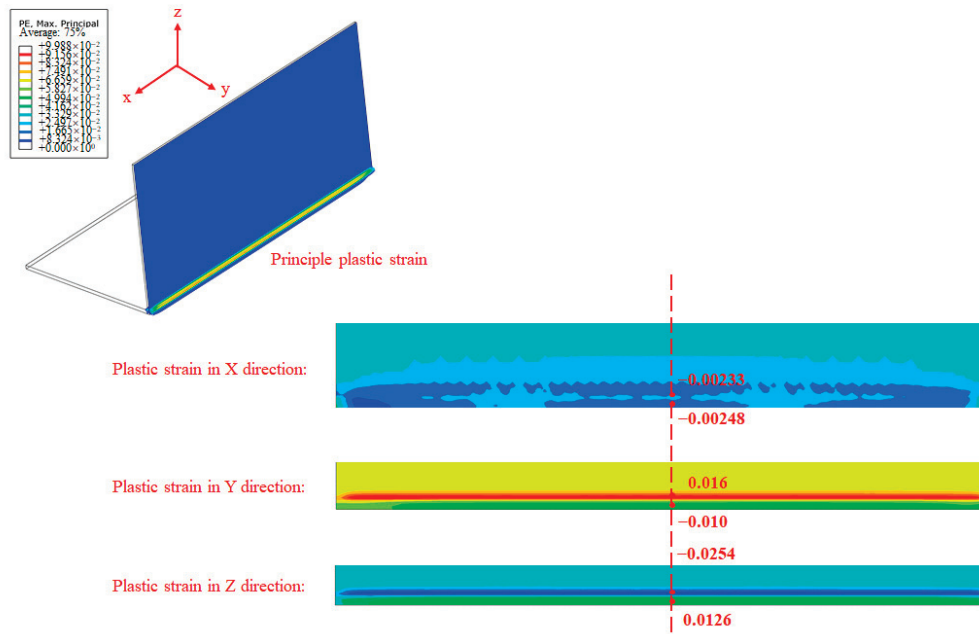


Figure 9. Diagram of the plastic strain distribution in the molten pool.

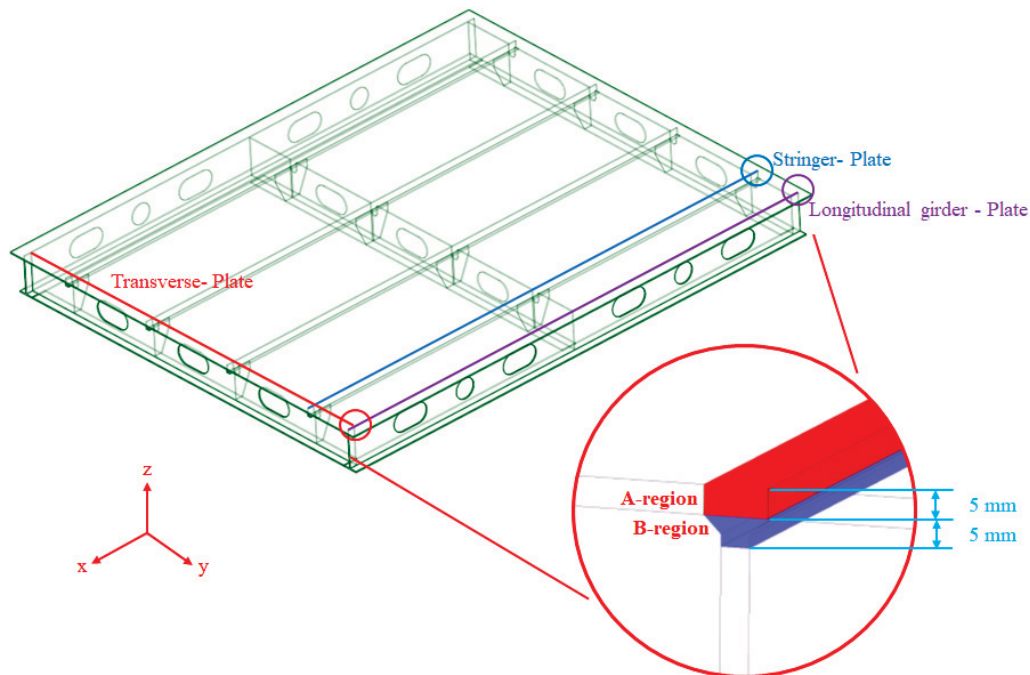


Figure 10. Diagram of the partitioning approach for the welding joints.

After determining the partitioning of the welding joint and the inherent deformations to be imposed, a solid element model was established. The deformation and stress distribution of the plated grillage were simulated using the ABAQUS (2022) software on an 8-core (4.5 GHz) 16-threaded PC with 64 GB of RAM. The computational time for a single model was about 55 min.

The model is meshed using hexahedral elements, with the element type C3D8R, and the number of elements and nodes were 665,548 and 779,472, respectively. Encryption of the mesh was conducted in the heat-affected region around the welding joints. The boundary conditions were set not to unduly restrict the degrees of freedom of the structure and to ensure that the constraints of the specimen were close to the actual assembly welding

process [34]. The detailed boundary condition settings were shown in Figure 11. The stress distribution and deformation of the plated grillage were shown in Figures 12 and 13.

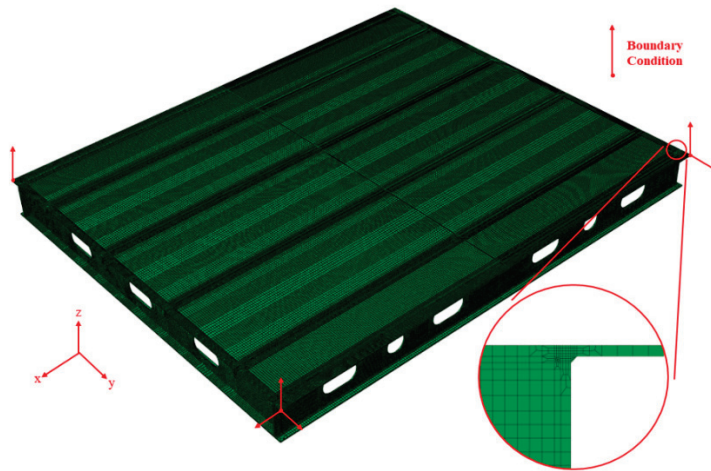


Figure 11. Diagram of solid element model of the plated grillage.

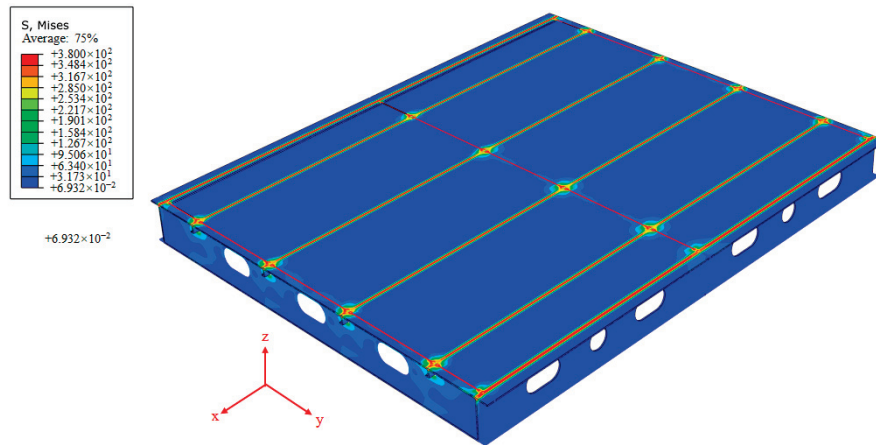


Figure 12. Contour diagram of stress distribution in the plated grillage.

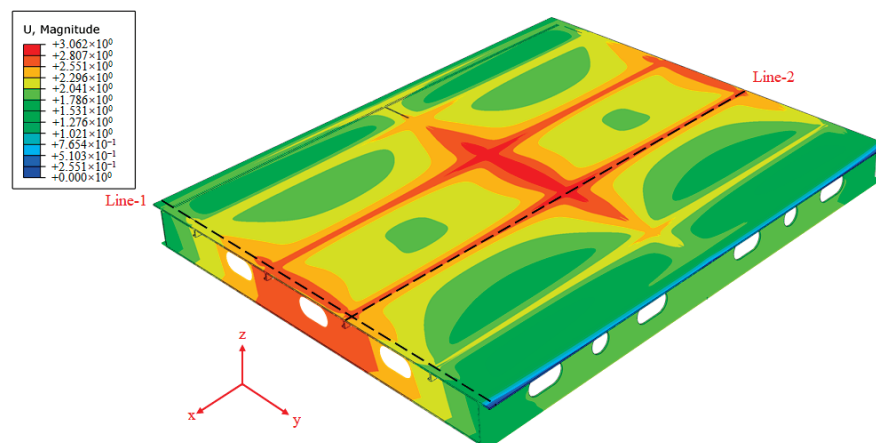


Figure 13. Contour diagram of deformation distribution in the plated grillage.

As shown in Figures 12 and 13, there are relatively high stresses in the welding joint region that reached the yield strength of the material, causing plastic deformation. A certain number of high-stress areas were also presented in the transverse due to the shrinkage of the welding joint of the longitudinal stringers. For the deformation condition, the maximum deformation on the plated grillage is about 3.1 mm. To verify the agreement

between the simulated and measured deformation of the plated grillage, two monitoring lines (as Figure 13) were set up above the plate region connecting with the longitudinal stringer and transverse, and the simulated and measured vertical deformation values were compared. Deformation values were measured using an XTDIC 3D optical acquisition system with a measurement accuracy of ± 0.1 mm. Data collection was performed at 100 mm intervals in the length and width directions of the specimen, with the number of collection points being 20 and 16, respectively (as shown in Figure 14). Comparison results were shown in Figure 15.

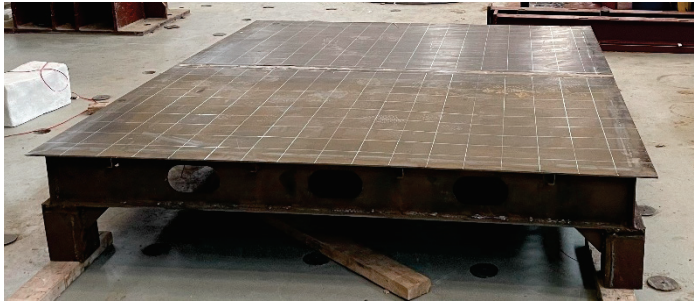


Figure 14. Diagram of the actual plated grillage specimen.

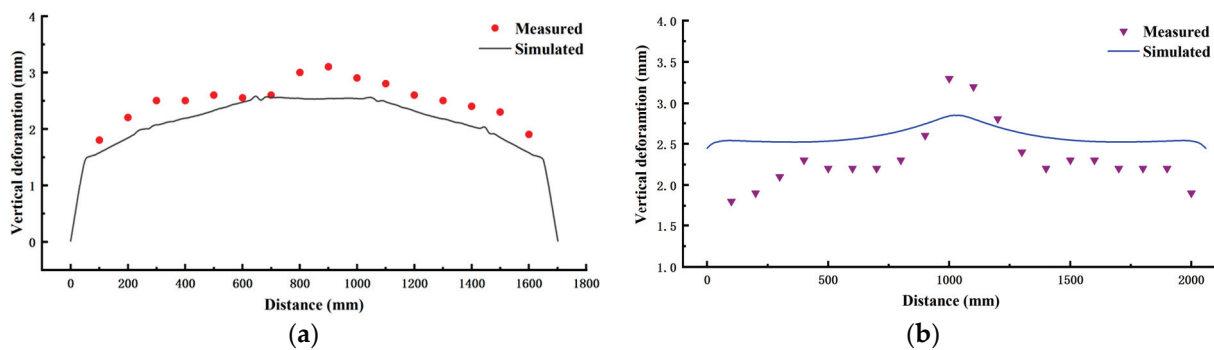


Figure 15. Diagram of the deformation comparison results of monitoring lines. (a) Line 1 and (b) line 2.

As shown in Figure 15a,b, the deformation values of the monitoring lines fluctuated significantly at the welding joint. The monitoring line 1 direction presented a large amount of deformation due to the denser welding joint arrangement, and the maximum value of the simulated vertical deformation results reached 2.58 mm. The measured values corresponded well to the simulated values. The fluctuation of the deformation in the monitoring line 2 direction was small, with only a small peak around the welding joint region of the middle transverse. When the specimen was manufactured, there was a splice weld in the middle of the deck due to the limitations of the available plate sizes. Although the deck splicing was corrected after completion, it still resulted in a large error in the measurements at the center. Measured values away from the splice seam region were in good agreement with the simulated values.

4. Investigation of the Failure Buckling Behavior of Combined Opening Plated Grillage Considering the Initial Welding State

4.1. Numerical and Experimental Basis

The nonlinear finite element method was applied to calculate the failure behavior of the plated grillage. The welding deformation and residual stresses calculated in Section 3 were also applied as initial states to investigate their effect. On this basis, static load experiments were designed and performed, and the experimental results were compared with the simulated values to verify their reliability.

4.1.1. Simulation Settings

The nonlinear finite element calculation of the plated grillage was also conducted using ABAQUS (2022) software. The calculated welding deformations and stresses were applied as the initial load to the plated grillage model for further failure analysis. The meshes were divided in the same forms as used in the inherent strain calculation above, and the element type is C3D8R. The material parameters of the specimen were kept the same as mentioned in the welding simulation above. The Young's modulus of the material was 206 GPa and the yield strength was 380 MPa at 20 °C. The tangential modulus of the material was mostly taken as 5–15%, and there it was taken as 10% in combination with the results of the tensile experiments (as Figure 16). The specimen size for the tensile experiment was determined according to the Chinese national standard (GB-T 228.1-2021) [38].

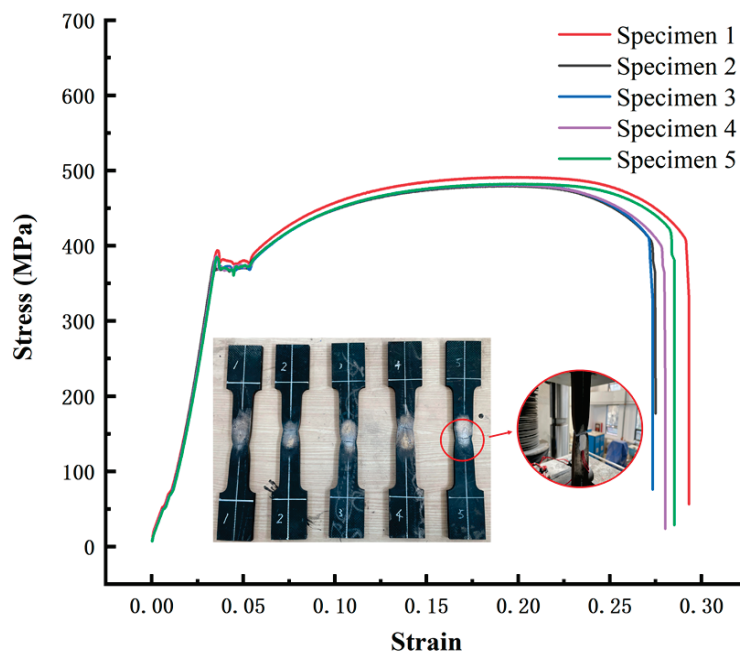


Figure 16. Diagram of the tensile experiment results.

In the previous experiments on the combined opening girders, it was found that the structure would fail in a complex manner such as instability and local buckling under the central vertical load applied from the plate [39]. Most of the existing plated grillage experiments focused on the failure behavior under longitudinal and transverse compressive loads, ignoring the effect of vertical bending loads from the plate [28,29]. Therefore, two loading conditions were designed. One was to load an area with a 200 mm width in the middle of the longitudinal direction. The other is to load an area with a 200 mm width in the middle of the transverse and longitudinal directions in combination. For the setting of boundary conditions, the main consideration is the support of the surrounding pillars for the plated grillage, so the position of the plated grillage and pillar connections were equivalently replaced by fixed constraints, and the failure behavior of the plated grillage under bending loads was mainly investigated. The loading method and boundary condition settings were shown in Figure 17.

In Figure 17, R_x , R_y , and R_z are the rotational degrees of freedom in the x -, y -, and z -axis; and U_x , U_y , and U_z are the translational degrees of freedom.

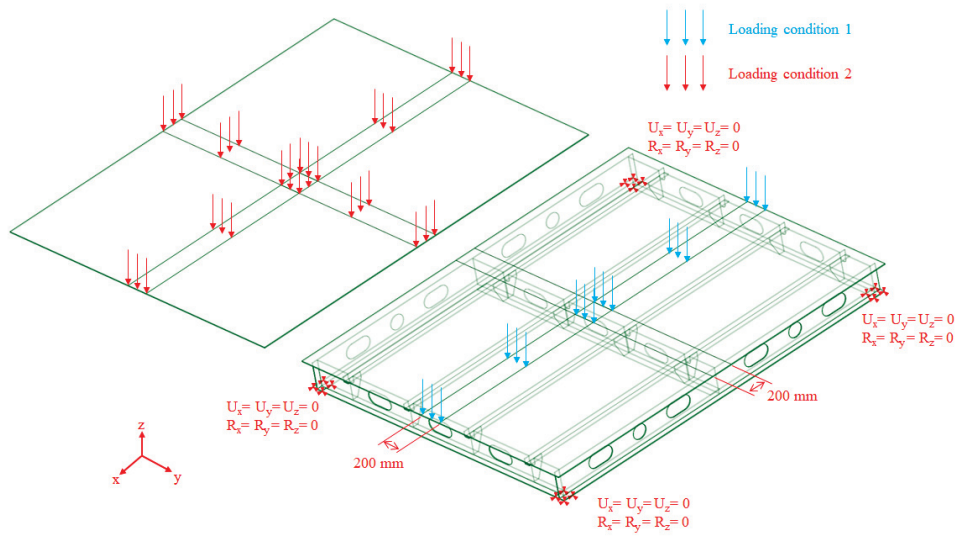


Figure 17. Diagram of the loading conditions and boundary condition settings.

4.1.2. Experiment Design

The vertical loading of the specimen was applied using a 4000 kN thrust servo actuator. To realize the two load conditions described in Section 4.1.1, the corresponding design of the distribution girders was performed. The experiment scenario was shown in Figure 18.

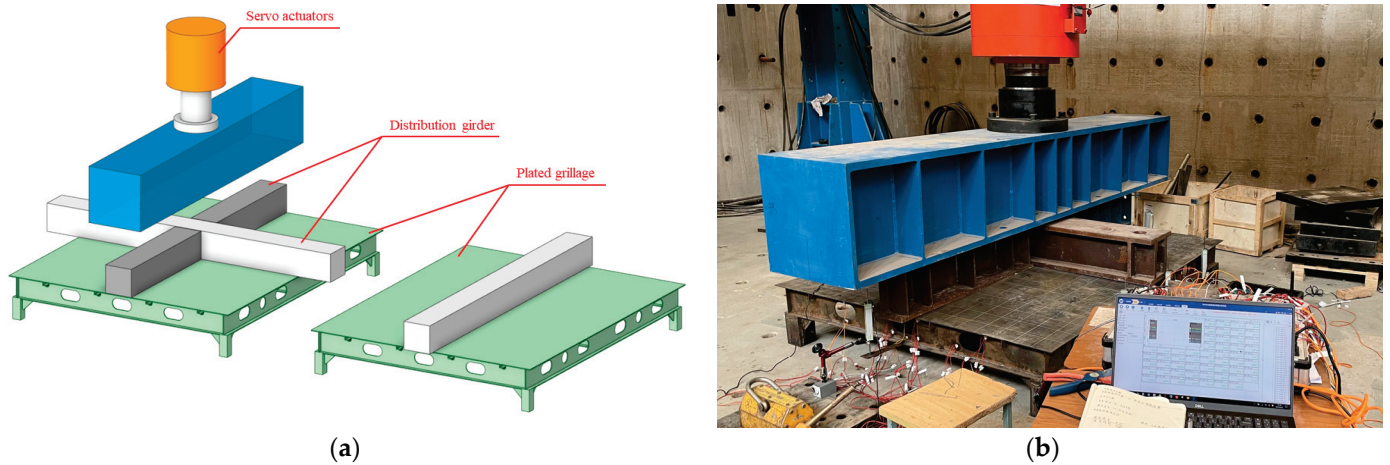


Figure 18. Diagram of the experimental scenario. (a) Schematic model and (b) physical scenes.

To collect the strain and stress in the plated grillage, displacement gauges were arranged at several typical points and strain gauges (rosettes) were attached to the steel surface. The type of displacement gauge was YWD and the measurement accuracy was ± 0.01 mm. The types of strain gauges and rosettes were BFH120-2CA-D150 and BFH120-2AA-R-P300, respectively, with sensitivity coefficients of $2.0 \pm 1\%$. The arrangement of displacement gauges and strain gauges (rosettes) was shown in Figure 19.

During the conducting of the experiment, the force-controlled loading method was first applied at the linear deformation stage of the plated grillage, with 20 kN per step. When approaching the plastic deformation stage, the displacement-controlled loading method was applied, with 1 mm per step. Then the loading was maintained until the value of the vertical pressure recorded by the pressure transducer dropped to 50% of the peak load value. At this point, the loading was stopped, and the experiment was completed.

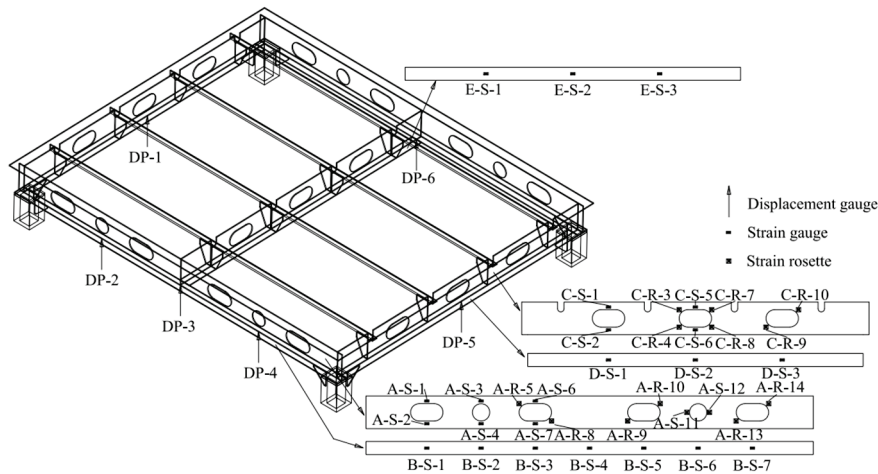


Figure 19. Diagram of arrangement of measurement gauges.

4.2. Experiment and Simulation Results

4.2.1. Load-Displacement Relationships

Due to the differences in the loading method, the two specimens exhibited different deformation and failure behaviors. In this section, the values of three displacement gauges (DP-1, DP-3, and DP-5) were selected for comparison. Meanwhile, the vertical displacement values of the plate center point obtained from simulation and measurement (considering the initial welding state) were compared. The load-displacement curves were shown in Figure 20.

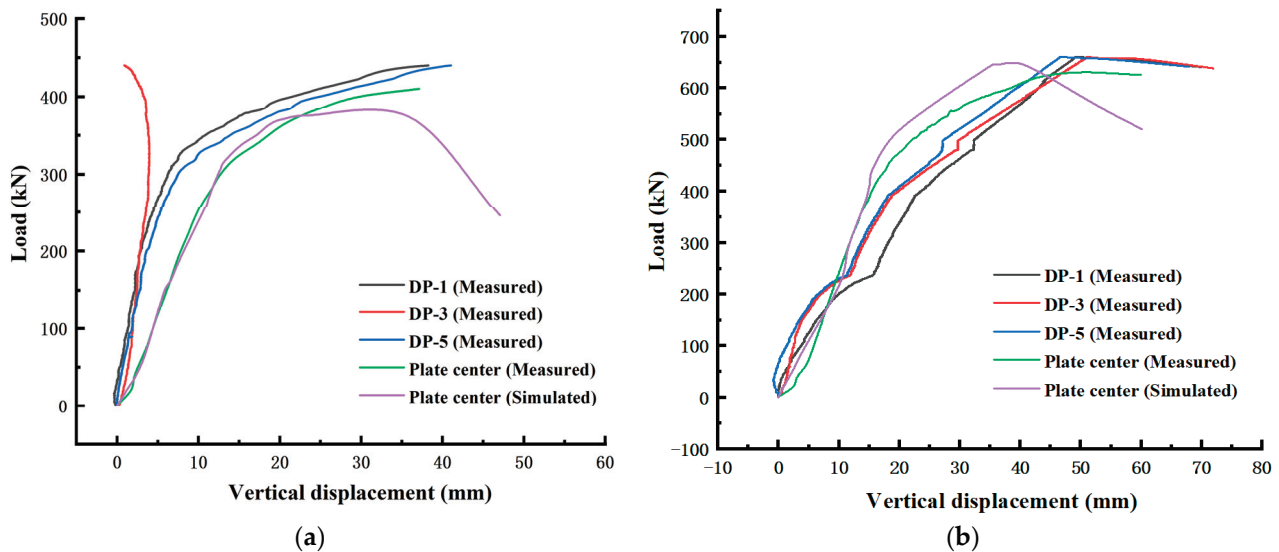


Figure 20. Diagram of the load-displacement curve. (a) Condition 1 and (b) condition 2.

For the first loading condition, the maximum load that the structure could bear was about 430 kN. As shown in Figure 20a, as the loading region was an area of 200 mm width in the middle of the longitudinal direction, and the middle of the front and back transverse exhibited similar displacement regulations under vertical loads. The location of the middle point of the plate also showed a similar displacement regulation to the DP-1 and DP-3 regions under the action of the distribution girder. For the values measured from DP-1, since the region was far away from the loading region, less vertical displacement occurred. As for the whole loading process, when the load is in the range of 0–300 kN, the structure was in the stage of elastic deformation. With the increase in the external load, the vertical displacements of all four measurement points increased linearly. When the

external load exceeded 300 kN, the structure gradually entered the plastic deformation stage. The slopes of the load-displacement curves at points DP-1, DP-5, and the plate center became smaller, and the structure experienced a distinct vertical displacement. The position of measurement point DP-3 was rotated around the X-axis due to the middle deformation of the structure; the position of the displacement gauge was shifted, so the measured displacement values appeared to decrease with the increase in the external load. Next, as the external load further increased to reach the critical load of the structure, there was a sudden loss of the load-bearing capacity of the structure. The loading device stopped loading for protection, so the measured curve in the figure terminated at the highest load point and did not record the post-buckling characteristics of the structure. The simulated load-displacement curve of the plate middle point fitted well with the measured curves and reflected a certain degree of post-buckling characteristics.

For the second loading condition, the maximum load the structure could bear was about 660 kN, with an improvement of 53.49% compared to the loading condition 1. As shown in Figure 20b, as the loading region was a combined area of 200 mm in the middle of both the longitudinal and transverse directions, the middle of the left and right longitudinal girders as well as front and back transverse exhibited similar displacement regulations for the load applied by the distribution girder. For the loading process, when the load is in the range of 0–600 kN, the structure was in the stage of elastic deformation. As the external load increased, there was a slow linear increase in the vertical displacement of the structure. When the external load reached 600 kN, the structure entered the plastic deformation stage, the rise of the load slowed down, and the structure suffered distinct vertical displacement. However, differing from the failure characteristics of the sudden loss of the load-bearing capacity of the structure in loading condition 1, in loading condition 2, when the structure entered into the plastic deformation stage, the external load remained stable with the distinct increase in displacement. The external load on the structure gradually decreased only when the vertical displacement exceeded 50 mm. The structure exhibited a considerable load-bearing capacity in the post-buckling phase.

4.2.2. Deformation

Combining the results of experiments and simulations, the failure behaviors of the plated grillage were investigated. The deformation of the structure after failure was shown in Figure 21.

As shown in Figure 21, the simulated failure behavior of the plate, longitudinal girders, and transverses were basically consistent with the results obtained from the experiment. For loading condition 1, owing to the vertical load applied to the area of a 200 mm width in the middle of the longitudinal direction, the plated grillage exhibited a bending deformation in the middle region. The transverses, which bore most of the loads as the main transverse component, suffered distinct deformation. In contrast, the section modulus at the opening region in the transverses was greatly reduced due to the presence of lumbar round openings. The bending load experienced a transmission discontinuity at the lumbar round opening regions, and the transmission path was shifted. Stress concentrations appeared at the corners of openings, resulting in a localized plastic hinge failure phenomenon (as Figure 21a). In addition, under this loading method, the longitudinal girders were arranged parallel to the direction of the loading region in the center, and no distinct deformation occurred. Only the rotation of the whole longitudinal girders along the X-axis direction occurred with the increase in the external load.

For loading condition 2, owing to the loading region being an area of a 200 mm width in the middle of the transverse and longitudinal directions, both longitudinal girders and transverses participated in load-bearing. Subjected to external loads, the middle regions of the left and right longitudinal girders as well as the front and back transverses exhibited distinct vertical deformations. Due to the action of distributing girders, the vertical deformation in the middle region of these major longitudinal and transverse components was consistent. Similarly, the longitudinal girders and transverses showed a localized plastic

hinge failure around the lumbar round opening regions in loading condition 1. As for the longer length of the longitudinal girders, the plastic hinge deformation phenomena were more pronounced at the lumbar round openings closer to the middle loading region than at the distal end.

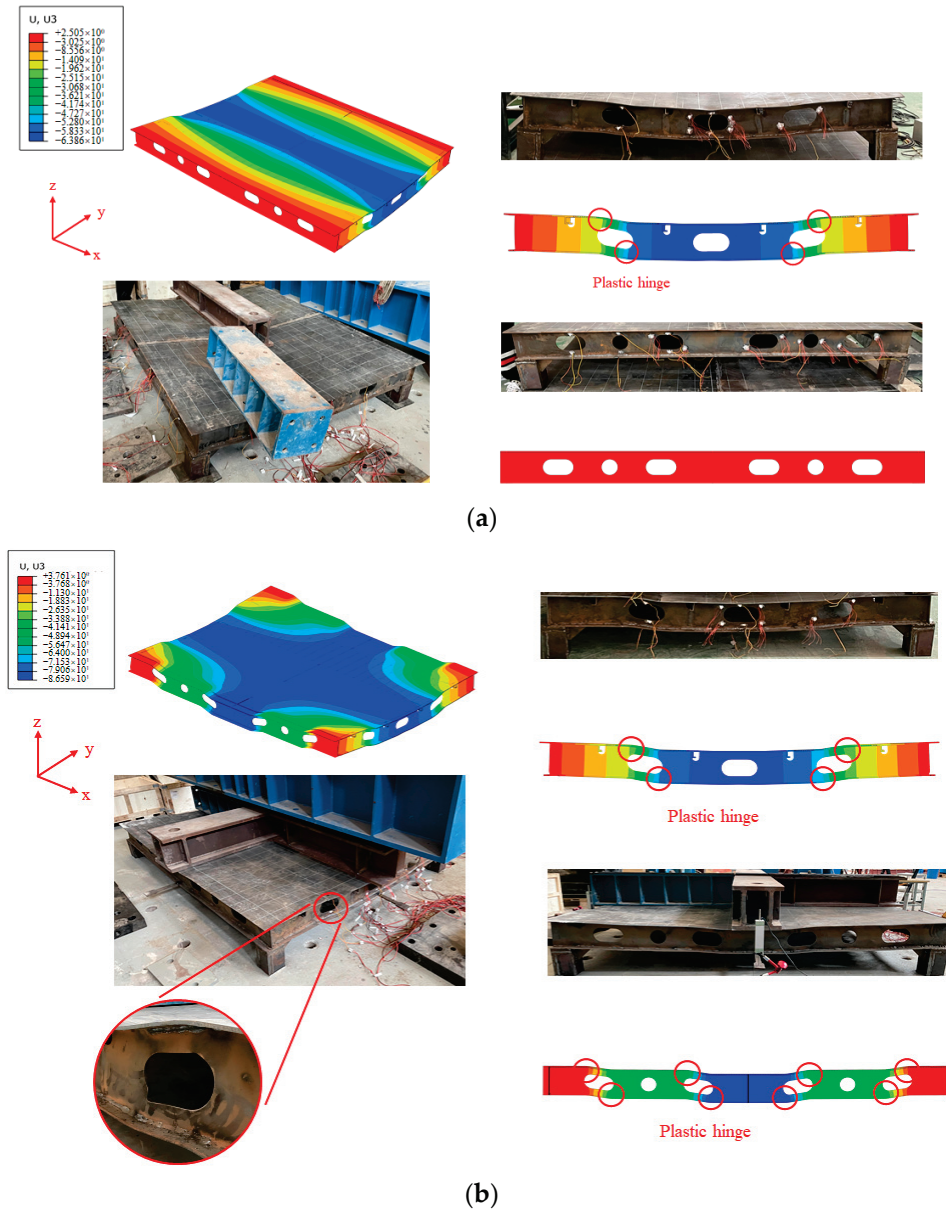


Figure 21. Diagram of the failure behaviors. (a) Condition 1 and (b) condition 2.

For two loading conditions, the central region of the transverses presented a significant vertical deformation. In contrast, in the static load experiments of the girders in the literature 39, the girder structures presented a phenomenon of instability due to the occurrence of lateral slip in the middle region. It could be seen that the inclusion of the ribs in the plated grillages could limit the local instability phenomenon of the structure and improve the load-bearing characteristics.

In addition, to investigate the deformation regulations of the plate of the plated grillages and verify the agreement between the simulation and experiment results, the coordinates of the measuring points on the deformed plate surface were measured by the XTDIC 3D optical acquisition system. The curved surfaces were formed based on the fitting of coordinate values and compared with the results of simulations. The comparisons were shown in Figure 22.

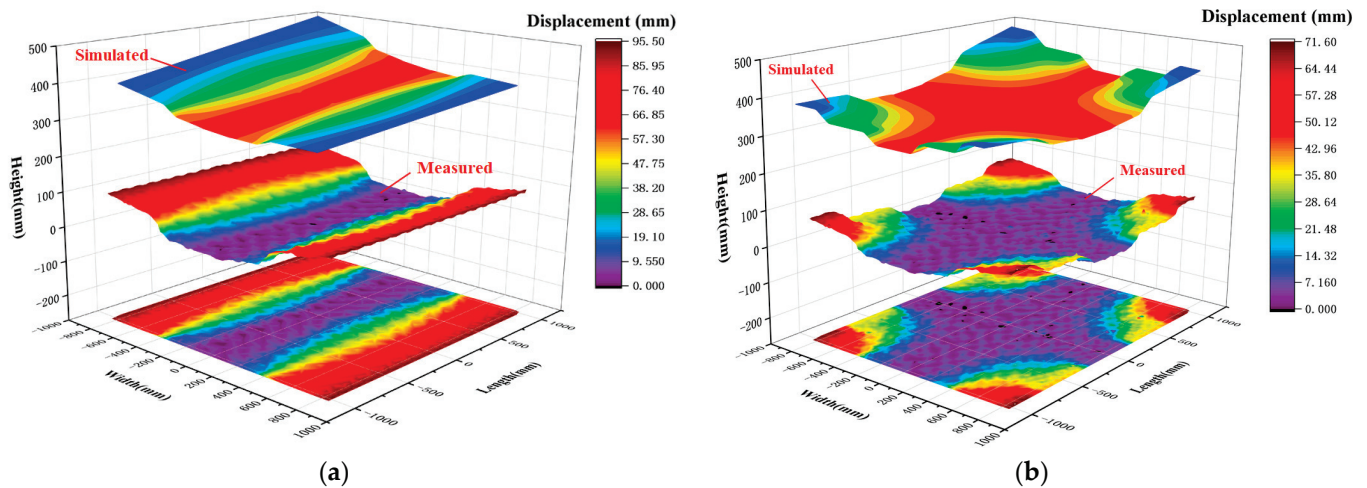


Figure 22. Diagram of the simulated and measured comparison results of plate deformation. (a) Condition 1 and (b) condition 2.

As shown in Figure 22, the curved surface shapes obtained from the fitting of the measured data were in good agreement with the results of the simulation. For loading condition 1, the plate exhibited a buckling band similar to a hyperbolic symmetrical arrangement on both sides of the longitudinal center of the plate (as Figure 22a). The entire plate showed a vertical deformation approximating the shape of a cylinder in the region between the two lumbar round openings along the X-axis. As for loading condition 2, there are significant differences in the deformation patterns of the plate. The vertical displacement values of the central region in both the X and Y axes were approximately equivalent. The plate exhibited a gradual increase in vertical displacement from the corners to the middle region in both the X and Y directions. Both directions showed a localized bending of the plate above the lumbar round openings, resulting in a significant change in vertical displacement on both sides of the bend. From the corners toward the center of the plate, buckling bands similar to the inverse function distribution appeared (as Figure 22b).

4.2.3. Stress and Strain

Due to the differences in the loading conditions, there were significant differences in stress distribution in the plate frame at failure. For loading condition 1, the stress level on the longitudinal girders was low and there was no significant stress concentration around the opening regions. The transverses, due to bearing most of the external loads, showed a high-stress region in the middle of the stiffener flange and a significant stress concentration around the opening. For loading condition 2, both the longitudinal girders and transverses were involved in bearing the external loads, so the stress levels were higher in the middle of the stiffener flange. Meanwhile, there were stress concentrations at the corners of the lumbar round openings. The stress distribution was shown in Figure 23.

Referring to the stress distribution, it could be seen that due to the difference in the loading conditions, there was a significant difference in the stress level in the middle of the stiffener flange of the longitudinal girders and transverses. The strain values measured experimentally at this location are discussed next. The values on the surface of specimens during the experiment could be measured by strain gauges (rosettes). The selected measuring points were numbered B-S-4 and D-S-2. The strain values of the measuring points were shown in Figure 24.

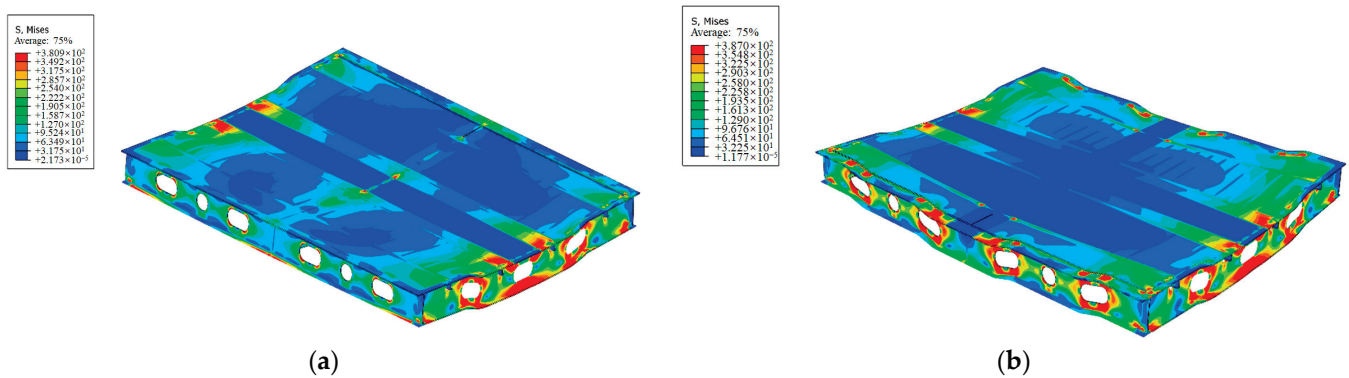


Figure 23. Contour diagram of stress distribution in the plated grillage. (a) Condition 1 and (b) condition 2.

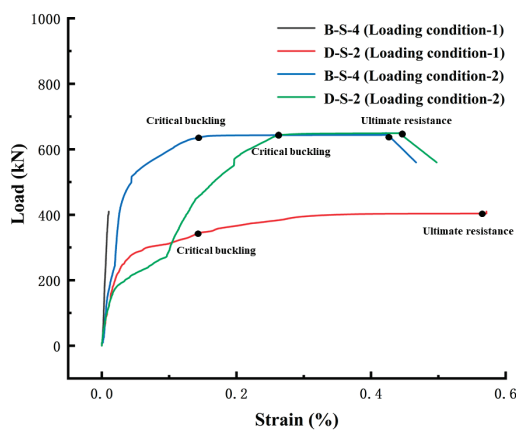


Figure 24. Diagram of the strain values of the measuring points.

As shown in Figure 24, for loading condition 1, since the main load in this loading method was borne by the transverses, the strain measured from the location of the measuring point on the longitudinal girder was small, less than 0.1%. The strain measured from the location of the measuring point on the transverse was large, especially when the external load reached 370 kN, the strain value increased rapidly, and the structure lost its load-bearing capacity. For loading condition 2, since both the longitudinal girders as well as the transverses were involved in load-bearing, the strain values measured at the locations of the two measurement points were large. In the pre-loading stage, the transverses were subjected to a large load, producing a more pronounced strain. When the external load came to 450 kN, the longitudinal girders also participated in the load-bearing, and the strain growth rate increased significantly. In this case, the increased rate of strain at the measuring point on the longitudinal girder was larger than that on the transverse. When the external load reached 610 kN, the strain values at the two measuring points gradually reached the same and increased rapidly; at this time, the structure gradually lost its bearing capacity.

4.3. Influence of the Initial Welding State on Structural Load-Bearing Behaviors

The degree of congruence between the numerical results was verified above. With the use of the inherent strain method, the distribution of stresses and deformations of specimens generated during the welding process was also simulated. In this section, the load-bearing behavior of the ideal model, the model considering initial deformation, and the model considering initial deformation and stresses were compared for two loading conditions using numerical means. The load-displacement relationship curves of the plate center point for these conditions were shown in Figure 25.

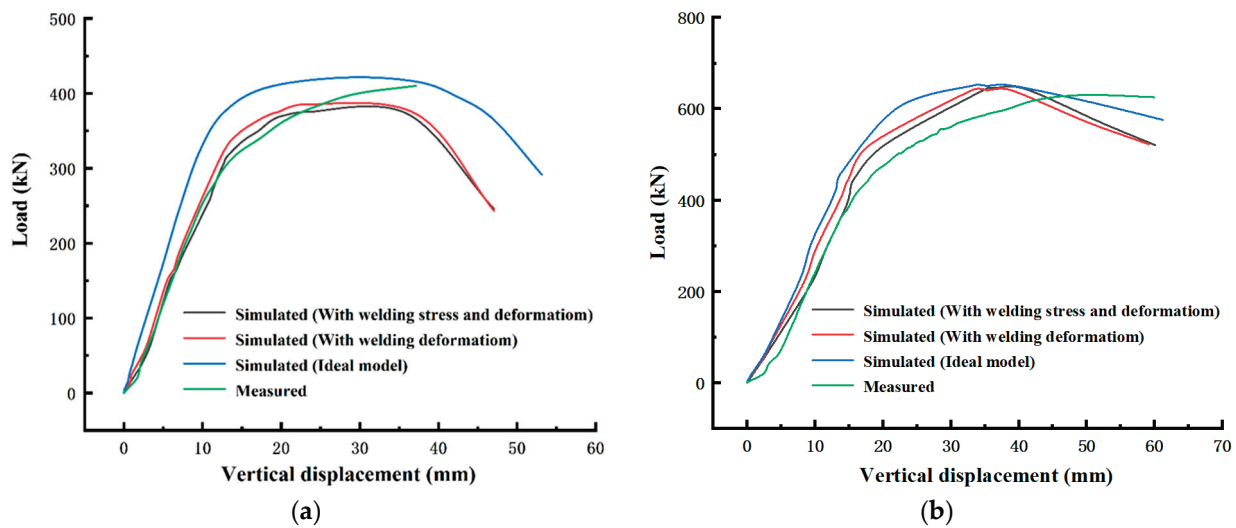


Figure 25. Diagram of the load-displacement curve considering the initial welding state.

As shown in the Figure 25, considering the initial welding state of the specimens, the ultimate load-bearing capacity decreased slightly compared to the ideal model and had a better fit to the experimental load-displacement curves. For loading condition 1 in the Figure 25a, the curve considering welding residual stresses and deformations almost coincided with that only considering deformation. For the welding parameters used in this structure, the welding residual stresses had almost no effect on the deformation structure in the elastic phase. As the external load continued to increase, the residual stresses made the structure more susceptible for plastic deformation while leading to a 2.5% reduction in the ultimate load-bearing capacity. Next, comparing the curves considering only the welding deformation with those of the ideal model, it was evident that the welding deformation has a significant effect on the deformation behavior of the structure. In the elastic phase, the welding deformation enabled the structure to generate a larger vertical displacement under the same external load and entered the plastic phase earlier. The ultimate load-bearing capacity showed a decrease of 8.6%.

For loading condition 2 in the Figure 25b, compared to welding deformation, the effect of welding residual stresses on the deformation behavior of the structure remained insignificant. The load-displacement curves for the two cases nearly overlapped, with only a small difference in the position at which the structure entered the plastic phase. Next, compared with the curve of the ideal model, the presence of welding deformation accelerated the vertical displacement of the structure in the elastic phase, causing the structure to yield earlier. However, the influence degree was less than that in loading condition 1. When the structure failed, the ultimate load-bearing capacity of the structure was essentially the same in all three cases. It may be seen that under this loading mode, the welding residual stresses and deformations caused by the welding parameters used in this specimen had basically no effect on the ultimate load-bearing capacity of the structure.

5. Conclusions

A simulation method that partitions the welding joint region to apply different inherent strains was used to calculate the initial welding deformation and stress based on actual welding parameters. Then the results obtained from the calculations were used as an initial state for nonlinear finite element calculations, and the deformation behaviors of the structure were simulated and compared with the results of the static load experiments.

The main findings are as follows:

- (1) The method of partitioning the welding joint region to apply different inherent strains could enable a more accurate prediction of welding deformation and stress distribution in the plated grillages. Referring to its simulation results, the welding deformation

of the plated grillage was mainly angular deformation caused by welding shrinkage at the connection of the longitudinal stringers, longitudinal girders, and transverses with the plate.

- (2) For the plated grillage, when the loads were carried by a combination of longitudinal and transverse components, the structure had a high ultimate load-bearing capacity. When the structure reached the ultimate load-bearing limit, the structure still had a considerable load-bearing capacity in the post-buckling phase. For loading conditions in which the load was applied to the components in a single direction, the ultimate load-bearing capacity of the structure was greatly reduced. And there was a phenomenon that the overall structure lost its load-bearing capacity due to localized structural failures.
- (3) The degree of influence of welding deformation and stress on the structural load-bearing behavior was related to the method of applying the external loads. When the vertical load was carried by a combination of longitudinal and transverse components, the overall load-bearing state of the structure was more stable, and the influence of welding factors on the ultimate load-bearing capacity of the structure was small. When the transverse members were mainly involved in load-bearing, the structure suffered from localized deflections of the whole longitudinal components. Welding factors would exacerbate this phenomenon, resulting in a significant reduction in the overall ultimate load-bearing capacity.
- (4) Higher stiffener web heights could exacerbate the instability of the structure under vertical loading, and lateral movement of the central parts of the structure occurred, leading to localized instability. Meanwhile, the plastic hinge failure behavior at the corners of the lumbar round openings was associated with a loss of the structural load-bearing capacity. The addition of ribs could improve the local failure behavior of the structure and increase the load-carrying capacity.

Author Contributions: Conceptualization, C.C., H.Z. and Z.L. (Ziqiu Li); methodology, C.C. and Z.L. (Zhengda Lv); software, C.C. and Z.L. (Zhengda Lv); investigation, C.C. and Z.L. (Ziqiu Li); resources, H.Z.; writing—original draft preparation, C.C.; writing—review and editing, C.C. and H.Z.; visualization, H.Z.; supervision, H.Z.; project administration, H.Z.; funding acquisition, H.Z. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Natural Science Foundation of China (52371323) and the Postgraduate Research & Practice Innovation Program of Jiangsu Province (KYCX22_3838).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data used to support the findings of this study are available from the corresponding author upon request.

Acknowledgments: The authors would like to acknowledge the financial support provided by the National Natural Science Foundation of China (52371323) and the Postgraduate Research and Practice Innovation Program of Jiangsu Province (KYCX22-3838), and also acknowledge Shanghai Waigaoqiao Shipbuilding Co., Ltd. and China Shipbuilding Cruise Technology Development Co. Ltd. for their support in conducting relevant experiments.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Shen, W.; Zhao, Y.; Li, L.; Qiu, Y.; Liu, E. Assessment of residual ultimate strength on plate girders with web opening under compression-bending loadings. *Ocean Eng.* **2019**, *187*, 106147. [CrossRef]
2. Zhao, Y.; Yan, R.; Wang, H. Experimental and numerical investigations on plate girders with perforated web under axial compression and bending moment. *Thin-Walled Struct.* **2015**, *97*, 199–206.
3. Liu, X.; Guo, Z.; Bai, D.; Yuan, C. Study on the mechanical properties and defect detection of low alloy steel weldments for large cruise ships. *Ocean Eng.* **2022**, *258*, 111815. [CrossRef]
4. Brown, C.J.; Yettram, A.L. The elastic stability of square perforated plates under combinations of bending, shear and direct load. *Thin-Walled Struct.* **1986**, *4*, 239–246. [CrossRef]

5. Shakerley, T.; Brown, C. Elastic buckling of plates with eccentrically positioned rectangular perforations. *Int. J. Mech. Sci.* **1996**, *38*, 825–838. [CrossRef]
6. Brown, C. Elastic buckling of perforated plates subjected to concentrated loads. *Comput. Struct.* **1990**, *36*, 1103–1109. [CrossRef]
7. El-Sawy, K.M.; Nazmy, A.S.; Martini, M.I. Elasto-plastic buckling of perforated plates under uniaxial compression. *Thin-Walled Struct.* **2004**, *42*, 1083–1101. [CrossRef]
8. Shanmugam, N.E.; Lian, V.T.; Thevendran, V. Finite element modelling of plate girders with web openings. *Thin-Walled Struct.* **2002**, *40*, 443–464. [CrossRef]
9. Shanmugam, N.E.; Thevendran, V.; Tan, Y.H. Design formula for axially compressed perforated plates. *Thin-Walled Struct.* **1999**, *4*, 1–20. [CrossRef]
10. Paik, J.K. Ultimate strength of perforated steel plates under combined biaxial compression and edge shear loads. *Thin-Walled Struct.* **2008**, *46*, 207–213. [CrossRef]
11. Paik, J.K. Ultimate strength of steel plates with a single circular hole under axial compressive loading along short edges. *Ships Offshore Struct.* **2007**, *2*, 355–360. [CrossRef]
12. Paik, J.K. Ultimate strength of perforated steel plates under edge shear loading. *Thin-Walled Struct.* **2007**, *45*, 301–306. [CrossRef]
13. Saad-Eldeen, S.; Garbatov, Y.; Guedes Soares, C. Strength assessment of steel plates subjected to compressive load and dent deformation. *Struct. Infrastruct. Eng.* **2016**, *12*, 995–1011. [CrossRef]
14. Saad-Eldeen, S.; Garbatov, Y.; Soares, C.G. Experimental investigation on the residual strength of thin steel plates with a central elliptic opening and locked cracks. *Ocean Eng.* **2016**, *115*, 19–29. [CrossRef]
15. Saad-Eldeen, S.; Garbatov, Y.; Soares, C.G. Experimental strength analysis of steel plates with a large circular opening accounting for corrosion degradation and cracks subjected to compressive load along the short edges. *Mar. Struct.* **2016**, *48*, 52–67. [CrossRef]
16. Saad-Eldeen, S.; Garbatov, Y.; Soares, C.G. Experimental compressive strength analyses of high tensile steel thin-walled stiffened panels with a large lightening opening. *Thin-Walled Struct.* **2017**, *113*, 61–68. [CrossRef]
17. Saad-Eldeen, S.; Garbatov, Y.; Soares, C.G. Buckling collapse tests of deteriorated steel plates with multiple circular openings. *Ocean Eng.* **2019**, *172*, 523–530. [CrossRef]
18. Saad-Eldeen, S.; Garbatov, Y.; Soares, C.G. Experimental failure assessment of high tensile stiffened plates with openings. *Eng. Struct.* **2020**, *206*, 110–121. [CrossRef]
19. Han, X.; Liu, Z.; Xu, X.; Ge, C. Study of Stiffness Calculation Method on Reinforced Concrete Beam Strengthened with CFRP. *Adv. Mater. Res.* **2013**, *744*, 455–458. [CrossRef]
20. Yang, C.X.; Liu, Z.; Sun, L.P.; Li, J. Finite element analysis of shear behavior of reinforced concrete beam strengthened by high-strength steel wire mesh. *Appl. Mech. Mater.* **2014**, *482*, 15–19. [CrossRef]
21. Sun, L.P.; Liu, Z.; Yang, C.X.; Zhang, Z.F. Simulating the stiffness of RC beams strengthened by CFRP with secondary load by ANSYS. *Appl. Mech. Mater.* **2014**, *444*, 1062–1066. [CrossRef]
22. Wang, P.; Wang, X.; Ma, N. Vertical shear buckling capacity of web-posts in castellated steel beams with fillet corner hexagonal web openings. *Eng. Struct.* **2014**, *75*, 315–326. [CrossRef]
23. Zirakian, T.; Showkati, H. Distortional buckling of castellated beams. *J. Constr. Steel Res.* **2006**, *62*, 863–871. [CrossRef]
24. Trahair, N.S. Deformations of geometrically imperfect beams. *J. Struct. Div.* **1969**, *95*, 1475–1496. [CrossRef]
25. Zhao, Y.; Yan, R.; Wang, H. Experimental Studies of Failure Behavior and Strength of H Beams with Stiffened Web Openings under Compression and Bending Loads. *J. Ship Mech.* **2016**, *20*, 315–322.
26. Li, W.; Chen, Y.; Liu, X. Effect of continuous opening of web on the characteristics of long-span frame. *Ship Sci. Technol.* **2022**, *44*, 16–19.
27. Gan, J.; Shan, O.; Wu, W.; Lin, Y. Ultimate strength analysis of typical perforated high web frame structure in cruise ships. *Chin. J. Ship Res.* **2021**, *16*, 181–188. [CrossRef]
28. Liu, B.; Yao, X.; Lin, Y.; Wu, W.; Soares, C.G. Experimental and numerical analysis of ultimate compressive strength of long-span stiffened panels. *Ocean Eng.* **2021**, *237*, 109633. [CrossRef]
29. Liu, B.; Gao, L.; Ao, L.; Wu, W. Experimental and numerical analysis of ultimate compressive strength of stiffened panel with openings. *Ocean Eng.* **2021**, *220*, 108453. [CrossRef]
30. Qiu, Y.; Yan, R.; Wang, N.; Shen, W.; Xu, S.; Li, M.; Qin, K. Stress amplification effect and fatigue strength evaluation of marine thin plate welded structure considering welding deformation: Theoretical and experimental analysis. *Thin-Walled Struct.* **2023**, *188*, 110871. [CrossRef]
31. Zhou, H.; Zhang, Q.; Yi, B.; Wang, J. Hardness prediction based on microstructure evolution and residual stress evaluation during high tensile thick plate butt welding. *Int. J. Nav. Archit. Ocean. Eng.* **2020**, *12*, 146–156. [CrossRef]
32. Zhou, H.; Wang, J. Accurate FE Computation for Out-of-plane Welding Distortion Prediction of Fillet Welding with Considering Self-Constraint. *J. Ship Prod. Des.* **2019**, *35*, 317–327. [CrossRef]
33. Zhou, H.; Wang, J.; Zhang, H.; Liu, J.; Mo, Z. Prediction and mitigation of out-of-plane welding distortion of a typical block in fabrication of a semi-submersible lifting and disassembly platform. *Mar. Struct.* **2021**, *77*, 102964. [CrossRef]
34. Zhou, H.; Yi, B.; Niu, Y.; Wei, B.; Du, S.; Zhao, H.; Liu, J.; Wang, J. Application of efficient TEP FE computation on accurate fabrication of cylindrical leg structure of jack-up rig. *Ocean Eng.* **2020**, *196*, 106812. [CrossRef]
35. Wang, J.; Rashed, S.; Murakawa, H. FE Analysis of Buckling Behavior Caused by Welding in Thin Plates of High Tensile Strength Steel. *J. Mater. Eng. Perform.* **2014**, *23*, 4358–4365. [CrossRef]

36. Ueda, Y.; Murakawa, H.; Ma, N. *Welding Deformation and Residual Stress Prevention*; Elsevier Inc.: Amsterdam, The Netherlands, 2012.
37. Murakawa, H.; Deng, D.; Ma, N. Concept of inherent strain, inherent stress and inherent deformation and inherent force for prediction of welding distortion and residual stress. *JWRI Trans.* **2010**, *39*, 103–105.
38. *GB-T 228.1-2021*; Tensile Testing of Metallic Materials Part 1: Room Temperature Test Method. National Standardization Management Committee: Beijing, China, 2021.
39. Chen, C.; Zhou, H.; Lv, Z.; Ge, X.; Xu, X. Experimental and Numerical Investigation of the Buckling Behavior and Strength of Combined Opening Plate Girders in Passenger Ships. *Metals* **2023**, *13*, 1256. [CrossRef]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Experimental and Numerical Analysis of Ultimate Carrying Capacity of a Funnel Structure with Opening under Wind Pressure

Lei Ao ^{1,2}, Ziqi Ding ^{1,3}, Bin Liu ^{1,2,*}, Zhiyong Pei ^{1,2,*}, Qin Tang ⁴ and Weiguo Wu ^{1,2}

¹ Green & Smart River-Sea-Going Ship, Cruise and Yacht Research Center, Wuhan University of Technology, Wuhan 430063, China; lei_aowh@163.com (L.A.); dingziqi@whut.edu.cn (Z.D.); mailjt@163.com (W.W.)

² Hubei Province Engineering Research Center on Green & Smart River-Sea-Going Ship, Wuhan 430063, China

³ School of Naval Architecture, Ocean and Energy Power Engineering, Wuhan University of Technology, Wuhan 430063, China

⁴ School of Foreign Studies, Jiangnan University, Wuxi 214122, China; 8353510219@jiangnan.edu.cn

* Correspondence: liubin8502@whut.edu.cn (B.L.); zhiyong_pei@whut.edu.cn (Z.P.)

Abstract: This study presents experimental and finite element investigations on the ultimate strength of the funnel structure of a large passenger ship subjected to wind pressure. An experimental test is conducted using a similar model to analyze the failure characteristics of the funnel structure. The model is designed based on similar theories to simulate the progressive collapse behavior of an actual ship's funnel under wind load. Additionally, a simplified wind loading device is also developed to apply large wind loads. Practical insights are provided in the research to assess the opening's influence on reducing the ultimate strength of funnel structures when suffering wind pressure. Results represent the failure initiated at the edges of the large opening, with stress concentration primarily occurring at the stiffener end, showing good agreement with the simulated results performed using the finite element method. Furthermore, the effects of different parameters on the ultimate strength of the funnel structure are discussed by using the numerical method. This analysis provides an important guide for the design of funnel structures of passenger ships with openings.

Keywords: funnel structure with opening; scaled model test; wind load; ultimate strength

1. Introduction

During the extended operational period, ships and offshore structures face constant threats from harsh ocean environments, such as strong winds, high waves, and extreme events like typhoons. These environmental factors may lead to fatigue, damage, or even collapse of the structures. Under extreme load scenarios like typhoons, more severe marine climatic conditions can result in significant structural vibrations and displacements [1,2]. This poses potential safety risks to the overall integrity of towering structures such as funnels and wind turbines [3]. It has been reported that all the wind turbines operated by Okinawa Electric Power Company were extensively damaged by Typhoon Maemi [4].

Large passenger ships are very complex objects from the structural point of view [5]. They usually have full superstructures that provide a range of amenities, including the swimming pool, central hall and shopping malls. These facilities cater to the varied requirements of passengers for transportation and leisure activities throughout their time at sea. In terms of their principal dimensions, the total height of the superstructure is nearly the same as that of the ship hull, bringing new structural problems.

When a cruise ship navigates in open waters, it experiences more forceful wind pressures, leading to a notable reduction in its velocity and stability [6]. Wang et al. [7] investigated the load characteristic induced by the wind acting on the passenger ship's superstructure. They found that the extreme wind pressure could cause a large bending moment.

Furthermore, different openings are widely applied in the structural design of large passenger ships in order to accommodate specific layouts as well as minimize the overall weight of the ship. However, these openings can lead to structural issues such as local plate fracture, stress concentration and a significant reduction in the limit strength of ship structures. Several research studies have been conducted on the residual strength of passenger ship structures with various openings [8–11]. The collapse behaviour of panels under the action of both compression and lateral pressure have also been studied [12].

The nonlinear finite element method (NFEM) has been extensively employed in the numerical analysis of limit strength of ship structures because of the advantage of reducing cost, ease of implementation and provision of detailed information on the structural collapse mode. Morshedsolouk and Khedmati [13] evaluated the effectiveness of composite superstructures in contributing to the bending moment and observed that the superstructure's dimension significantly influenced the hull's failure mode. Romanoff et al. [14] conducted similar researches, noting a considerable increase in load distribution by the superstructure as the neutral axis moved close to the ship bottom.

Considering that the full-scale experiments are hard to implement and are of high cost, the scaled model tests have been developed to analyze ship structure failures. Ensuring a proper similarity law is crucial in these model tests [15], as it effectively guides the design of scaled models, enabling a suitable understanding of collapse behaviour and ultimate strength in real ships. To accurately reflect the nonlinear stage of progressive collapse in actual ships, it is important to analyze factors influencing nonlinear similarity during the testing. Benson et al. [16] conducted experimental studies on hull girders's post-ultimate strength behaviour using three scaled girders. Hirdaris et al. [17] provided a comprehensive review on examining design loads for plate structures based on model tests as well as theoretical methods, with particular consideration given to environmental loads. Quispe et al. [18] performed a pure bending test on a scaled hull girder, which revealed the plates between stiffeners failed first. Pei et al. [19] enhanced the law for nonlinear similarity design and utilized it in a test model with a large opening subjected to combined longitudinal bending and torsion.

Numerous studies have been conducted to analyze the ultimate strength of local structures and hull girders through experimental methods. However, the failure experiment was not carried out on the ship funnel structures suffering the wind load. The present researches mainly focus on the wind load evaluation acting on the whole superstructure or the whole ship structure. In their study, Kulkarni et al. [20] conducted a comparison between numerical analysis and wind tunnel experiment in order to analyze the load coefficients of superstructures with various wind angles. Chen et al. [21] computed the wind loads acting on a cruise ship considering different load components. The findings indicate significant variations in the load coefficients depending on the incoming wind's angles. Janssen et al. [22] carried out a computational analysis of wind forces acting on ships in order to explore how simplifying geometries affect load calculations. They computed wind forces for four simplified models ranging from rectangular girders to complex ship hulls. It was found that substantial reductions in complexity can result in notable discrepancies between calculated wind forces. In addition, most of the research objects corresponding to the wind load on funnel structures are funnel on land. Insufficient attention has been paid to investigating potential failures or accurately assessing the ultimate strength associated with the funnel design. Zhang [22] studied the failure process of an industrial funnel under different load conditions. Bhatt and Vasanwala [23] examined the ultimate strength of a funnel under monotonic load while considering the effects of various materials.

In this paper, the funnel structure with openings from a passenger ship is selected to analyze its collapse behaviour under wind load, experimentally and numerically. The research aims at determining the ultimate strength of the towering structure for improving the design and construction of the funnel in passenger ships. A scaled model is designed based on the nonlinear similar law. A new loading device is designed to simulate the

wind pressure, and the feasibility of this method is demonstrated. The scaled model of the funnel is tested for failure and the buckling collapse behaviour is examined. The load–displacement curves and structural collapse patterns derived from the experiment are compared with the results through the nonlinear finite element calculation. The impact of the opening on reducing the ultimate strength of the funnel structure is assessed. The ultimate strength of the real ship funnel is finally predicted by using this similar law and the influences of various parameters on the funnel’s ultimate strength are analyzed for optimum structural design.

2. Scaled Experiment

2.1. Similarity Law

Given the high cost and operational challenges associated with conducting real ship tests, it is crucial to develop a scaled model that can accurately replicate the collapse mode of a full-scale structures for ultimate strength research. Thus, a suitable similarity law is needed to guarantee the corresponding relationship between the scaled prototype and the real structure. Firstly, geometric similarity is widely used to calculate the overall size of the scaled model [24]. The funnel works as a thin-walled cantilever beam under wind forces. Based on the beam theory, the geometric scaling law can be expressed as follows:

$$\begin{cases} C_A = A_r/A_s = C_L C_t \\ C_I = I_r/I_s = C_L^3 C_t \\ Q_s = \frac{Q_r}{C_L C_t} \end{cases}, \quad (1)$$

where C represents the scaled ratio of the test model to the actual ship; A denotes the area of the segment; L denotes the length; t denotes the plate thickness; I represents the moment of inertia; Q refers to the ultimate wind force. Additionally, r and s , respectively, refer to the real ship and its corresponding model.

In addition to considering geometric and material similarity, it is important to make sure that the local strength is similar. The local plate strength of the scaled model may not accurately reflect the actual ship due to the limitations of linear geometry similarity design. Considering the nonlinear properties, the geometries of the local stiffened plates should be corrected based on their ultimate strength, respectively. The average ultimate stress of stiffened plates under uniaxial compression can be predicted by Equation (2) given by Zhang and Khan [25]:

$$\begin{cases} \frac{\sigma_u}{\sigma_y} = \frac{1}{\beta^{0.28} \sqrt{1+\lambda^{3.2}}} \\ \beta = (b/t) \cdot \sqrt{\sigma_y/E} \\ \lambda = (a/\pi) \cdot \sqrt{A_s/I_s} \cdot \sqrt{\sigma_y/E} \end{cases}. \quad (2)$$

σ_u is the ultimate strength; σ_y is the yield stress; β is the plate slenderness ratio; λ is the stiffener slenderness coefficient; b is the plate width between stiffeners; t is the plate thickness; E is the elastic modulus; a is the spacing between transverse frames; A_s and I_s are the area and inertia moment of the stiffener with attached plate, respectively.

It is obvious that the ultimate strength of local stiffened plates is determined by parameters β and λ . Therefore, the nonlinear strength criterion for the local stiffened panel under compression is determined by

$$\begin{cases} \beta_p = \beta_m \\ \lambda_p = \lambda_m \end{cases}, \quad (3)$$

p and m represent prototype and model, respectively.

2.2. Model Design

The actual funnel structure is shown in Figure 1. The max plate thickness of the actual funnel body is restricted to 5 mm for the purpose of weight control. The length ratio, denoted as C_L , is established as 15:1, taking into account factors such as model costing,

laboratory conditions, and fabrication process. Similarly, the thickness ratio, denoted as C_t , is taken as 5:3. Firstly, the initial dimensions of the scaled model are determined by geometric similarity. Then, the sizes of local stiffened plates are changed slightly to keep the same ultimate compressive strength. The details about the scaled model as well as the real structures are shown in Figures 2–5.

Table 1 presents a comparison of geometric parameters for typical sections. The difference between the scaled model and the actual prototype is below 5%. Comparison of the plating slenderness ratio β and stiffener flexibility coefficient λ is shown in Table 2. The difference ratio is also within 5%. Thus, the designed model can reflect the elastic plastic characteristics of the actual structure.

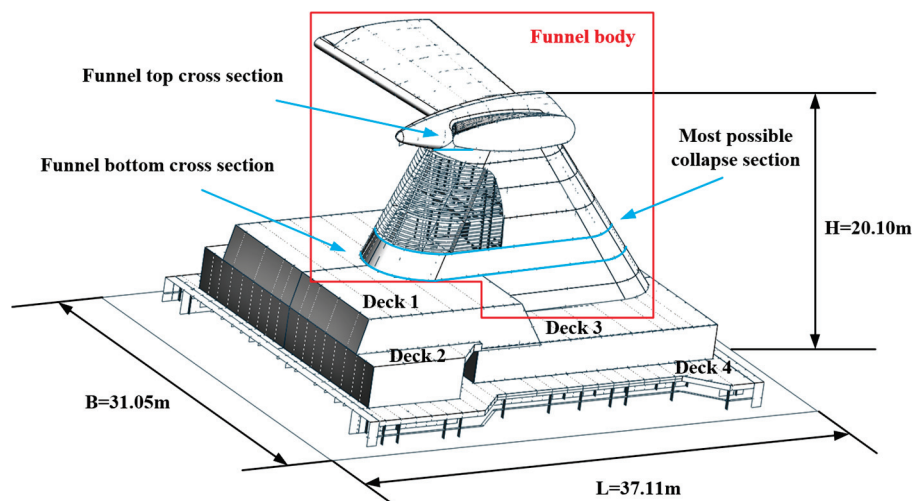


Figure 1. The main dimensions and compositions of funnel.

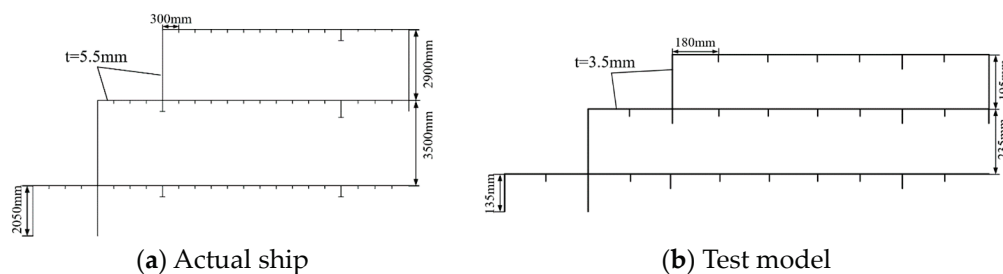


Figure 2. Comparison of cross-sections of Decks 1, 2 and 4.

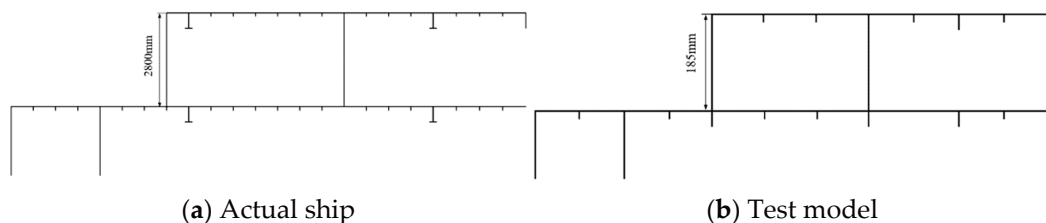


Figure 3. Comparison of cross-sections of Decks 3 and 4.

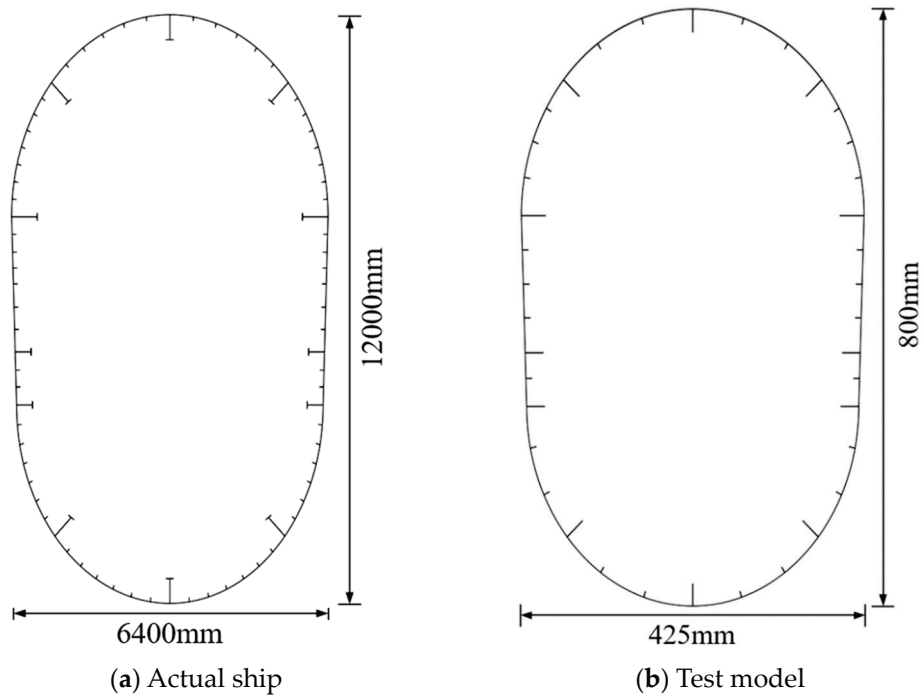


Figure 4. Dimensions of funnel top cross-section.

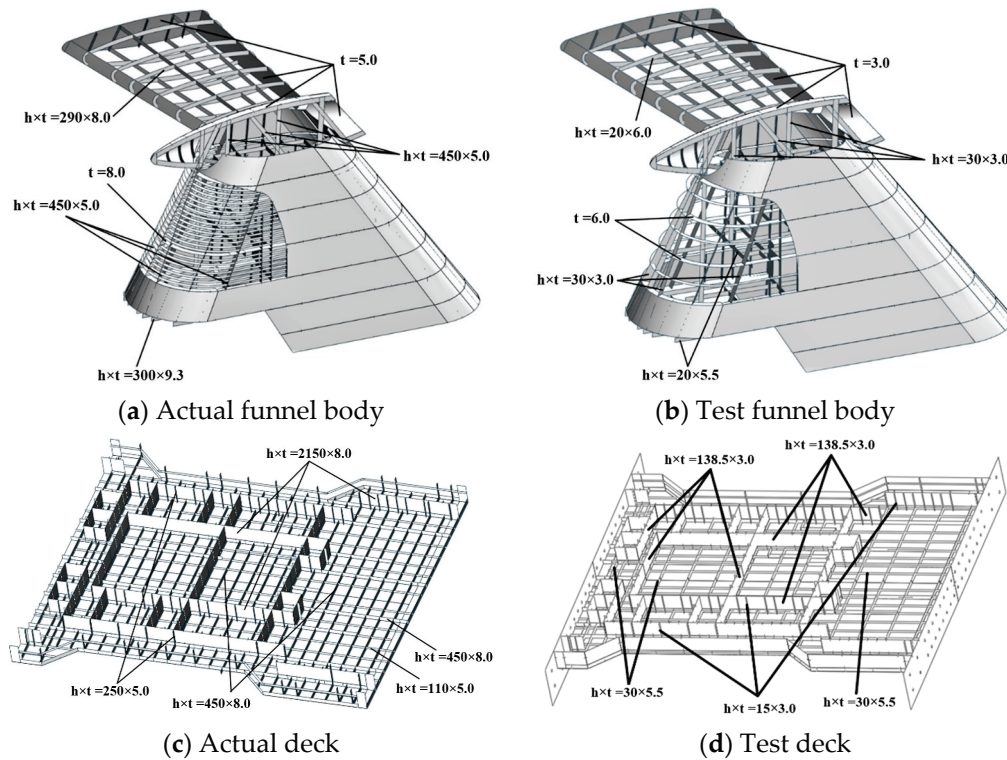


Figure 5. Geometry properties of plates and stiffeners.

The numerical calculation of the funnel structures is conducted using finite element software ABAQUS 2021. Figure 6 displays the details of FE models. The explicit code is utilized to address the structural post-buckling and collapse phenomenon and takes into account the large plastic deformation. The finite element model is generated using S3R and S4R type shell elements. S3R and S4R type shell elements are 3 and 4-node, stress/displacement shell elements with reduced integration and large-strain formulation

for tracking the large deformation [26]. The mesh size can have a significant effect on the uncertainty of the results obtained with finite element analysis. To obtain accurate numerical results, the meshes should be fine enough to well capture the yielding and local buckling process of the structure. However, finer meshes are accompanied by much greater consuming of computation efforts, which means that a balance between the mesh size and computing capability has to be made. Generally, more than 3 finite elements in the web of girders are generated, and the average element sizes of full-scale ship model for the refined parts and the scale model are 200 mm and 10 mm, respectively [27].

Table 1. Comparison of cross-section properties of the actual funnel to a scaled model.

Section	Parameters	Actual Ship	Similar Transformation	Model	Error
Deck 1	Section area (mm ²)	93,519.3	3740.8	3904.5	4.4%
	Position of neutral axis (mm)	357.0	23.8	24.5	2.8%
	Inertia moment about neutral axis (mm ⁴)	5.2×10^{10}	9.2×10^6	8.9×10^6	3.1%
Deck 2	Area (mm ²)	116,506.8	4660.3	4885.5	4.8%
	Position of neutral axis (mm)	311.4	20.8	20.5	1.2%
	Inertia moment about neutral axis (mm ⁴)	6.3×10^{10}	1.1×10^6	1.1×10^6	0.4%
Deck 3	Section area (mm ²)	104,539.5	4181.6	4353.5	4.1%
	Position of neutral axis (mm)	451.1	30.1	30.4	1.1%
	Inertia moment about neutral axis (mm ⁴)	6.1×10^{10}	1.1×10^6	1.1×10^6	4.6%
Deck 4	Section area (mm ²)	137,016.0	5480.6	5736.0	4.7%
	Position of neutral axis (mm)	202.5	13.5	13.6	0.5%
	Inertia moment about neutral axis (mm ⁴)	2.5×10^{10}	4.5×10^6	4.4×10^6	0.8%
Funnel bottom cross section	Section area (mm ²)	417,581.2	16,703.3	16,252.3	2.7%
	Moment of inertia (mm ⁴)	4.6×10^{12}	8.2×10^8	7.9×10^8	3.0%
Funnel top cross section	Section area (mm ²)	202,448.3	8097.9	7807.7	3.6%
	Moment of inertia (mm ⁴)	1.4×10^{12}	2.4×10^8	2.5×10^8	5.1%

Table 2. Comparison of nonlinear properties.

Members	β_p/β_m	λ_p/λ_m	$\frac{\sigma_{up}/\sigma_{ym}}{\sigma_{yp}/\sigma_{ym}}$ (Formula)	$\frac{\sigma_{up}/\sigma_{ym}}{\sigma_{yp}/\sigma_{ym}}$ (FEA)
The stiffened panel of deck1	2.23/2.01	0.87/0.88	0.62/0.64	0.64/0.65
The stiffened panel of funnel body	2.05/1.96	1.36/1.37	0.43/0.43	0.39/0.41

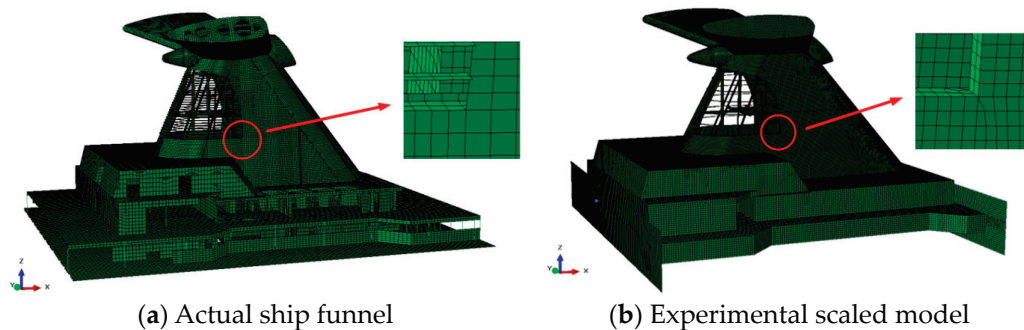


Figure 6. The numerical model of the funnel.

The loads and boundary conditions of the scaled model are shown in Figure 7. In this study, the uniform equivalent static wind pressure is adopted, and the surface pressure is applied on the windward side of the funnel structure to simulate the wind pressure. In engineering practice, wind action is often categorized into uniform wind and pulsating wind, and its load has a great relationship with waveform [28,29]. However, in the static

linear and nonlinear analyses, the actual stochastic dynamic wind load is generally modeled as a quasi-static pressure with gusty wind effects [30]. When the transverse load acts on the funnel, the funnel plate on the windward side is directly damaged due to excessive pressure. However, when the longitudinal load acts on the funnel, due to the influence of the large opening, the main components buckle under the bending moment of the ultimate wind load, and the ultimate strength of the funnel under the transverse load is much greater than the longitudinal load. Therefore, the longitudinal load is selected in the experimental analysis. For actual ship structure, A-A1 and B-B1 are supported by bulkheads, which are connected with multiple decks, so the fixed boundary is adopted; A-B and A1-B1 are supported by strong beams, so simply supported boundaries are adopted. The scale model is simplified, A-B and A1-B1 directly support the upper deck through the side plate, so the two sides are free and convenient for test implementation. The x symbol represents the longitudinal direction of the ship.

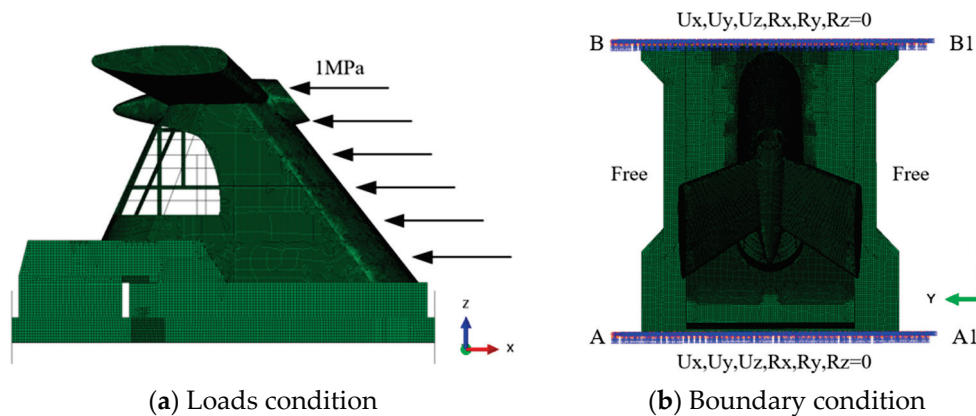


Figure 7. Loads and boundary conditions of the funnel.

The finite element calculation shows that the ultimate wind load of the prototype structure is 2.88×10^7 N, corresponding to the wind pressure of 0.38 MPa. According to the established similarity theory (Equation (1)), the ultimate load of the scaled model can be calculated as 1.15×10^6 N ($=2.88 \times 10^7 / 15 / 1.67$ N).

The ultimate strength of scaled model is 1.19×10^6 N, determined directly using the finite element method. Figure 8 shows load–deflection curves of the scaled model acquired through the application of the similar law and the finite element method. Observably, the ultimate strength of the scaled model, determined using the similarity formula, is comparable to the numerical value, with a discrepancy of 3.5%. The curves exhibit strong coherence within the elastic deformation. Nevertheless, it varies slightly during the plastic stage. In fact, some members are strengthened while some are removed during the model design, which causes an error in the structural carrying capacity. It can be found that under the same load, the displacement of the scale model determined by using the similar formula is smaller than that of the actually established scale model. It is mainly because the actually established scale model simplifies the internal components to some extent, resulting in the decrease in its stiffness.

The final collapse modes of both the prototype structure and scaled model are presented in Figure 9. It shows that the funnel primarily collapses in the corner panels of the opening. At first, the plates of the funnel body that are exposed to the wind encounter significant tension, with the highest levels of stress mostly focused on the welding joints. As the load increases, the funnel body slopes toward the opening. Thus, the panels situated on the windward side suffer the tensile force, and the ones on the leeward side undergo the compressive force. Subsequently, buckling takes place in the plate and stiffener components adjacent to the opening. As the load increases, large plastic deformation occurs in the vulnerable cross-section, resulting in the complete failure of the structure.

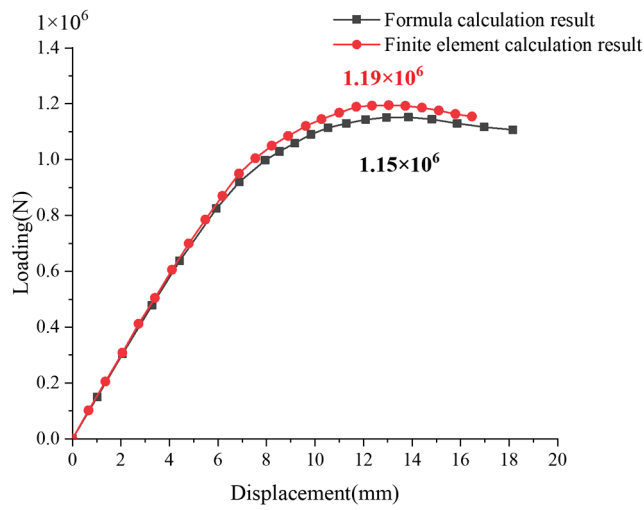


Figure 8. Load–displacement curves obtained by finite element analysis.

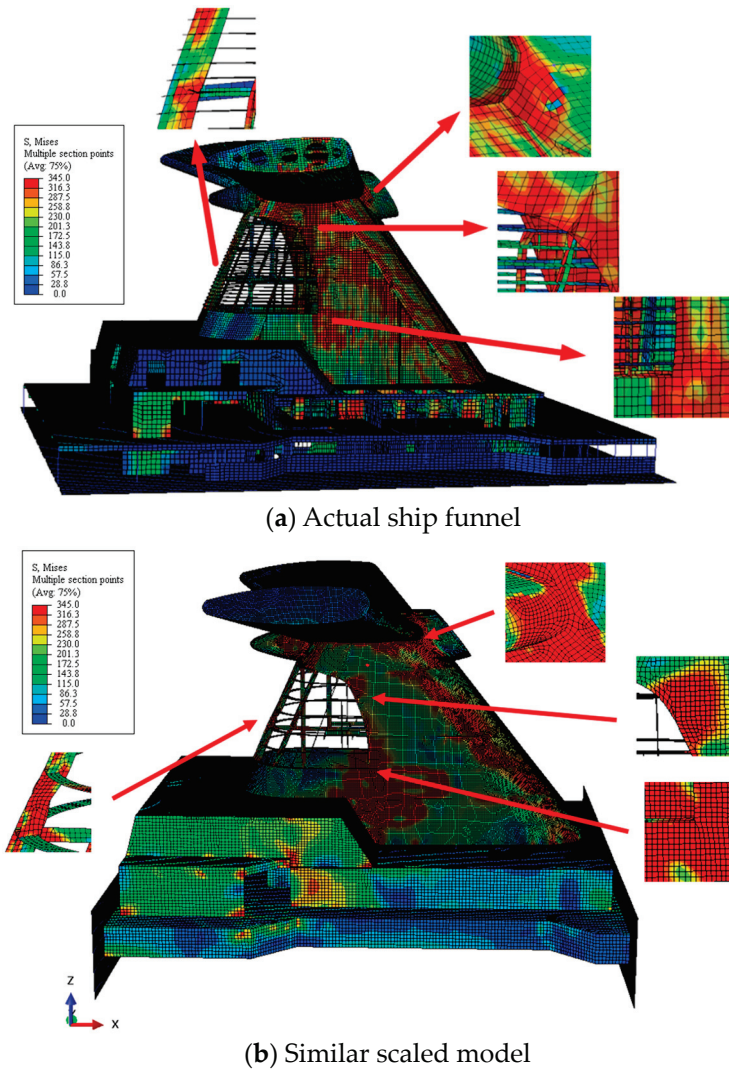


Figure 9. Stress distribution of two funnel models.

Figure 10 shows that both models have the largest Y-direction deformation at the corner of the opening. This illustrates that the failure of the whole funnel under wind load is mostly attributed to the local buckling of the stiffened plates at the opening, which

subsequently spreads to the adjacent members. Moreover, the finite element simulation confirms that the plastic deformation of both the full-scale ship structure and the test model is similar. Analyzing the loading and failure process of both models, it becomes evident that the scaled model precisely reflects the collapse characteristics of the real ship structure.

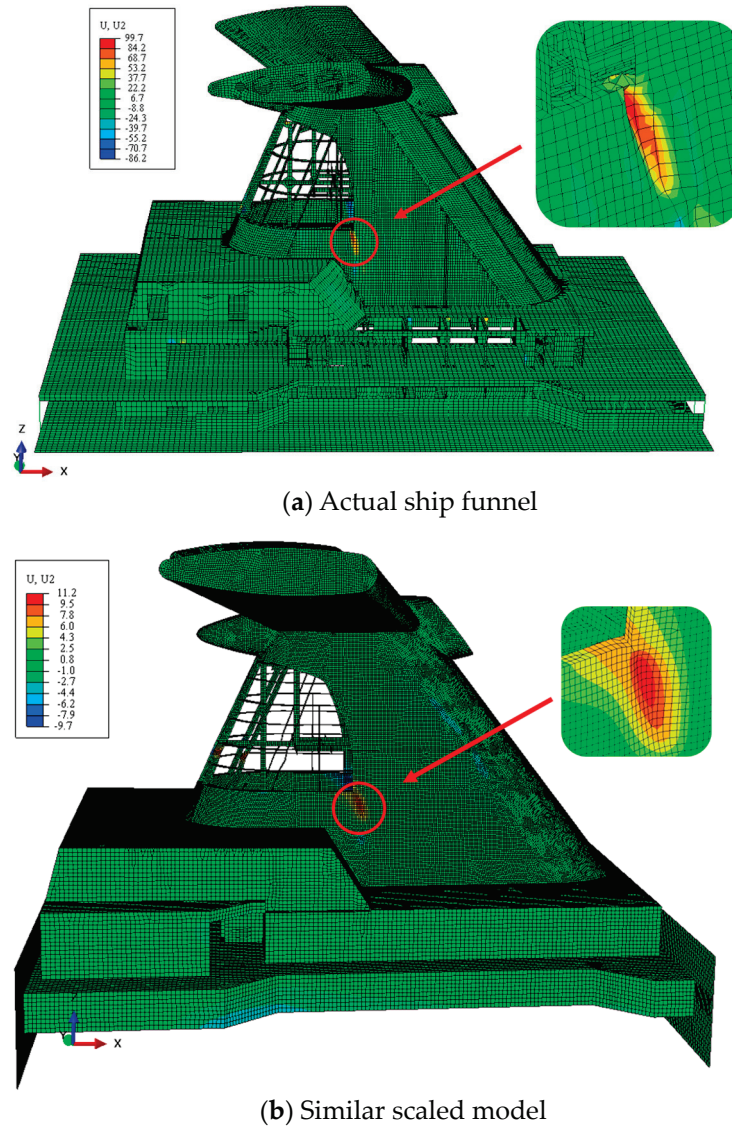


Figure 10. Y direction deformation of two funnel models.

In order to replicate wind loading during the test process accurately, a rigid bracket (loading support) is welded to the windward side of the scaled model as shown in Figure 11. This bracket serves the purpose of applying hydraulic pressure upon itself, which is then dispersed into many concentrated forces acting on the welded joints. The stiffness of the bracket is much larger than that of the funnel model to prevent the failure of the loading support in the testing process. Figure 12 illustrates a comparison of the ultimate strength of the test model with and without the loading support. The ultimate strength value of the scaled model with the bracket is 1.24×10^6 N, which is 4.2% larger than the unsupported model. This phenomenon is reasonable as the support is welded directly to the member's joints, enhancing the strength of local structures. Within the allowable error, the provided support reinforces the model indeed. In the elastic region for both models, the two load–displacement curves exhibit a good agreement. During the plastic deformation stage, there is a minor variation in the ultimate load.

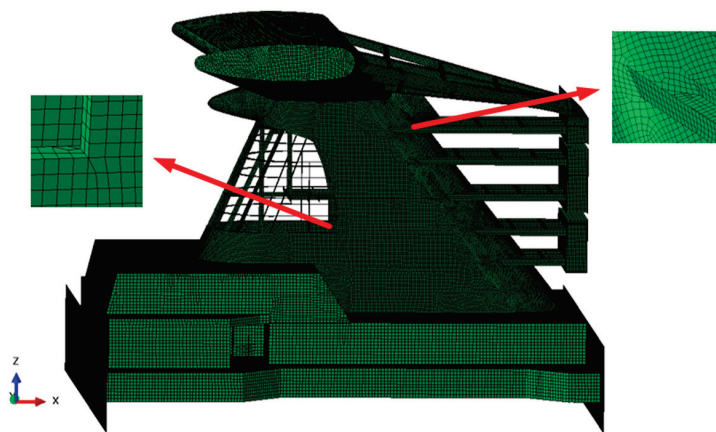


Figure 11. The FE model of the scaled model with support.

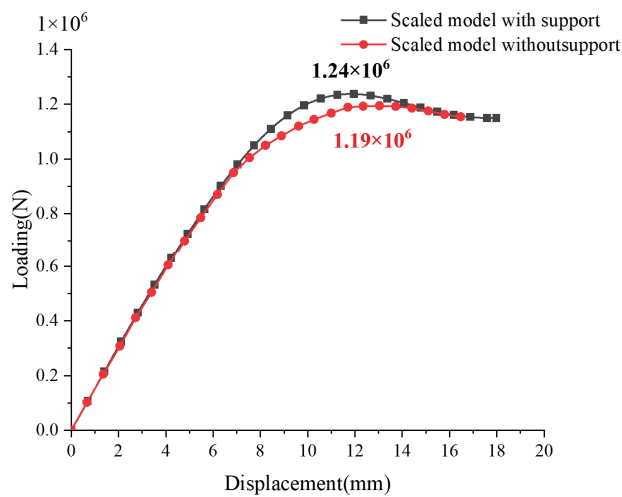


Figure 12. Load–displacement curves of scaled models with and without support.

Figure 13 shows the ultimate collapse mode of these two models. The collapse deformation of both models exhibits consistency at the limit state. The funnel bodies both are displaced backward under the action of forces exerted on their entire surfaces. The front panel undergoes tensile force. The plate and stiffener members near the opening suffer compression, leading to buckling and plastic deformation of local plates. The specimens mainly undergo global failure as a result of plastic buckling occurring at the opening cross-section. The funnel falls as a result of sudden and substantial deformation of the structural elements near the aperture, leading to a significant reduction in the external wind load.

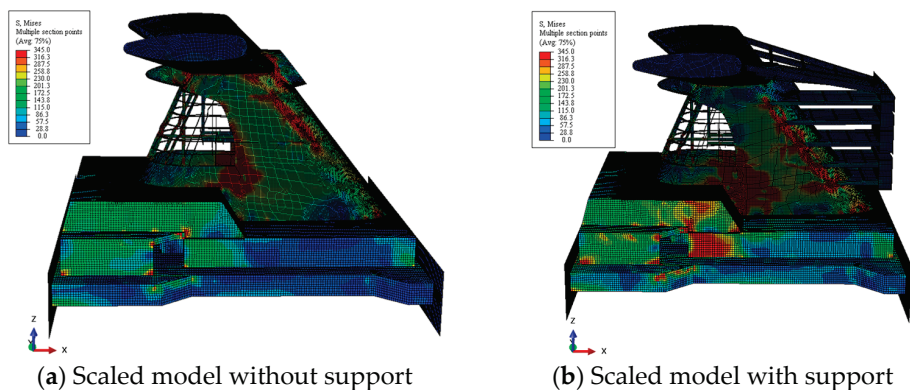


Figure 13. Numerical overall collapse of two scaled models.

2.3. Experiment Setup

The applied boundary and loading conditions in the scaled test are shown in Figure 14. Edges A-A1 and B-B1 are clamped and edges A-B and A1-B1 are free. Six hydraulic cylinders are utilized to provide external horizontal forces on one end of the loading support, and the concentrated forces are distributed as forces acting on the surface of the funnel body through the welded joints. Each hydraulic cylinder can apply a load of 300 kN and is synchronously controlled through an oil pump. At the same time, the pressure sensor is used to measure the applied force.

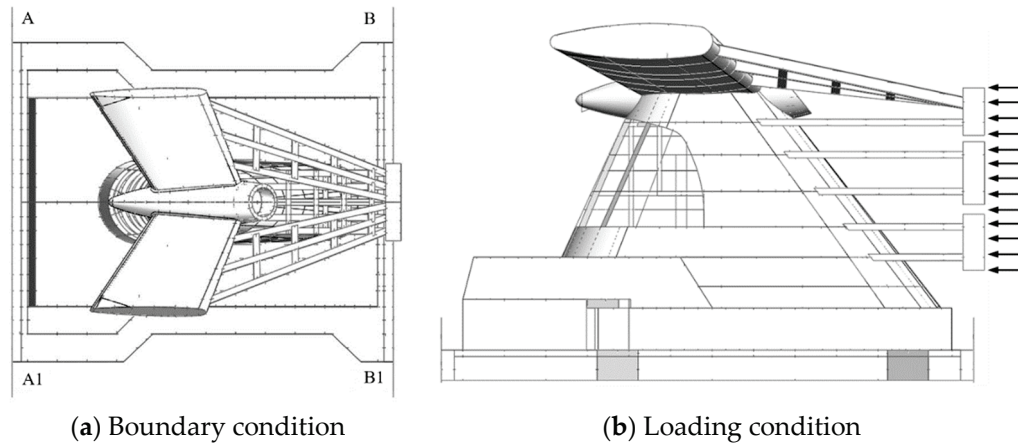


Figure 14. The boundary and loading conditions of the model test.

In order to prevent the frame from toppling over, enclosed supporting beams are established in the testing process and huge triangular cantilevers are employed to limit the x displacement of the closed frame (Figure 15). Series of hydraulic cylinders are utilized to apply significant horizontal wind load. These hydraulic cylinders are simultaneously controlled by an oil pump system. In addition, a load transducer is incorporated to record the input loads precisely.

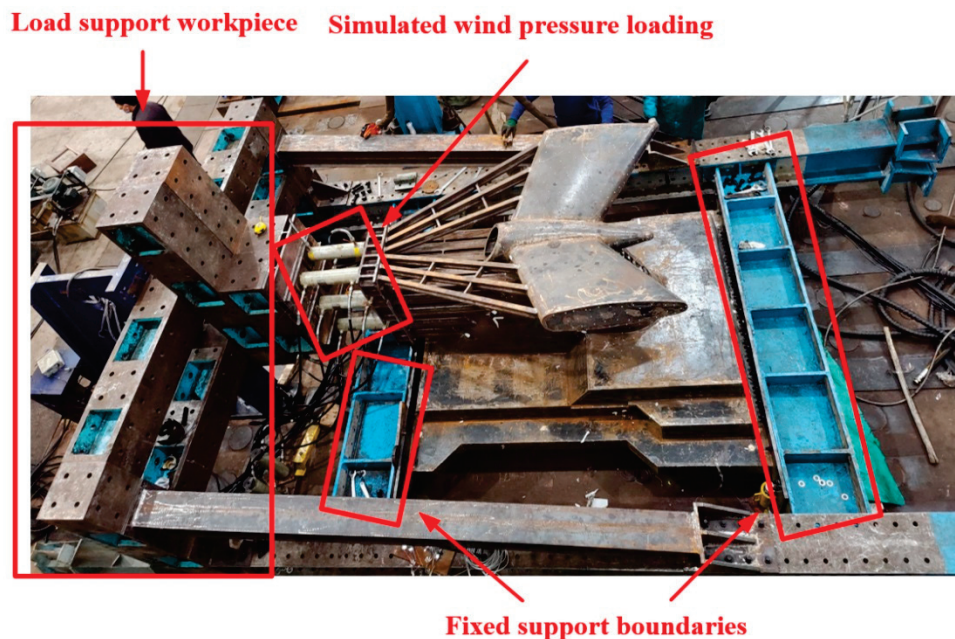


Figure 15. The collapse experiment setup.

The steps involved in carrying out the experiment are described in the following manner. (1) In the initial stage, several cycles of loading and unloading procedures are

carried out in order to partially release the residual stresses that were created in the welding process and reduce the gaps between the supporting beams. (2) Hydraulic cylinders are controlled to load the test specimen progressively and simultaneously until it collapses.

2.4. Tensile Test

The test model primarily consists of Q345 steel plates ranging in thickness from 3.0 mm to 5.0 mm. Its mass density is 7850 kg/m^3 , Young's modulus is 210 MPa, and the Poisson's ratio is 0.3. The tensile tests are carried out to acquire the material properties, as shown in Figure 16. The yield stresses of the steel materials, determined from the tensile results, are presented in Table 3.

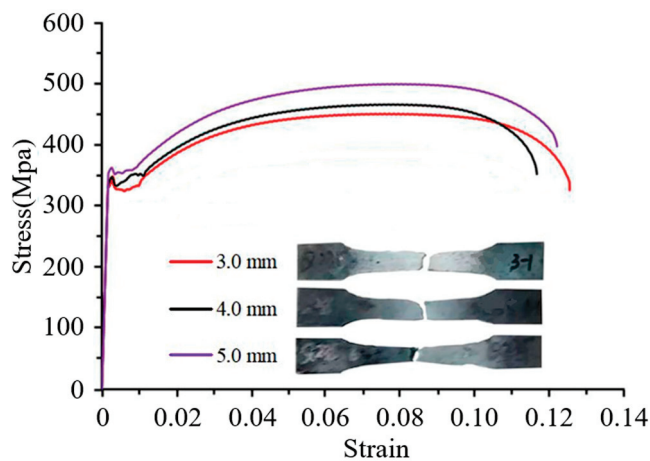


Figure 16. Stress–strain curve of test specimens.

Table 3. Yield stress of the steel materials.

Thickness	Yield Stress (MPa)
3 mm	335.4
4 mm	348.6
5 mm	359.4

2.5. Measuring Device Setup

Figures 17 and 18 illustrate the arrangement of strain gauges. These gauges are positioned in areas where large stress is concentrated, particularly at the region where welded joints buckle, areas with sudden and significant deformation, and the stiffeners of the opening. Prior to the experiment, finite element studies were performed to determine the locations of yield stresses for the test model.

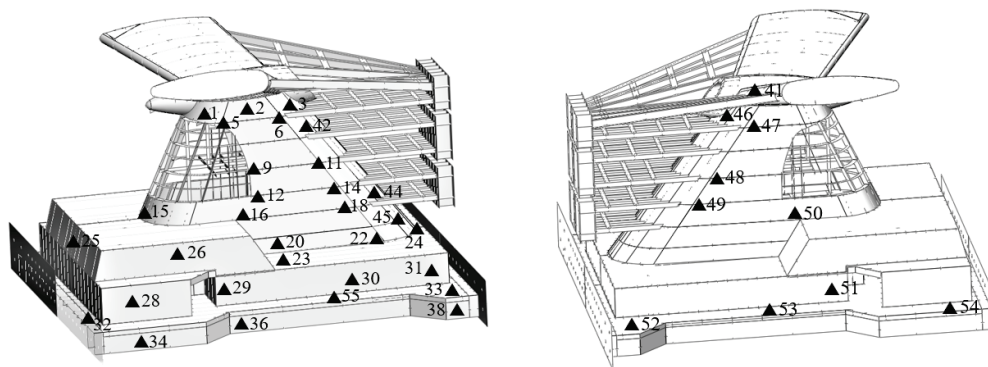


Figure 17. Layout of strain gauge on panel.

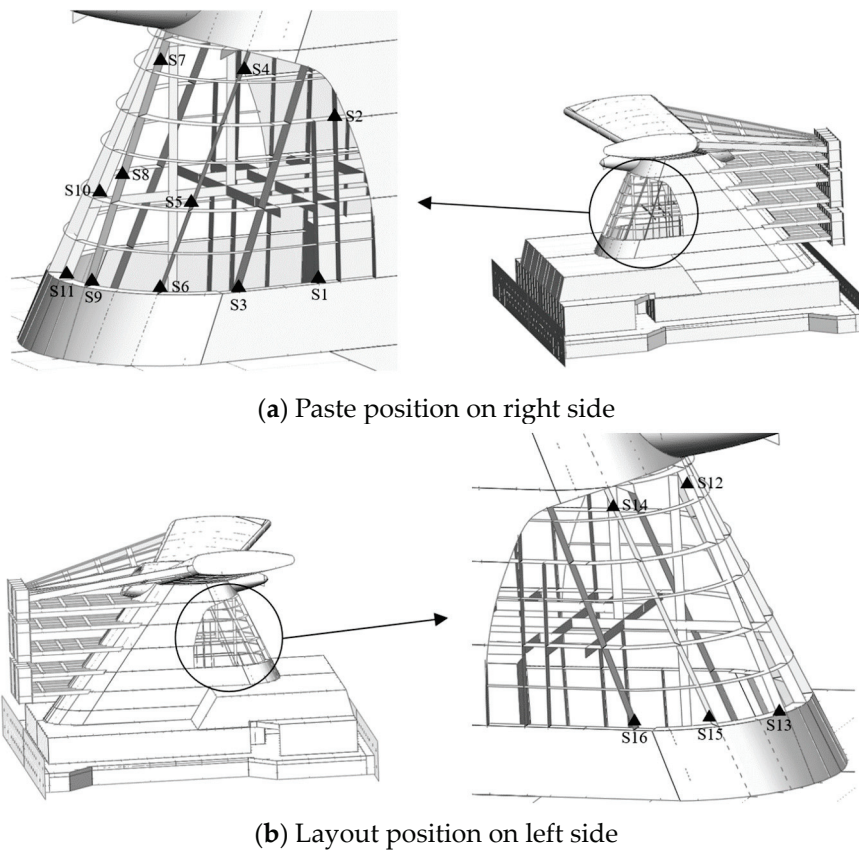


Figure 18. Layout of strain gauge on stiffeners.

The layout of the displacement transducers is shown in Figure 19. Measuring points D1, D2 and D4 measure the horizontal displacement of the specimen caused by horizontal load, and D3 measuring point measures the out-of-plane deformation of the deck.

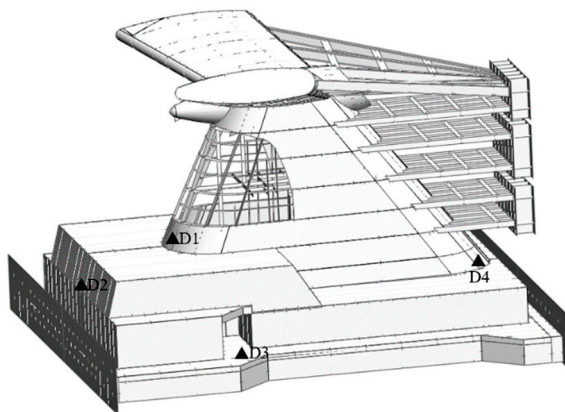


Figure 19. Layout of displacement transducers.

3. Experiment Results

3.1. Load–Displacement Relationship

Figure 20 displays the load–displacement curve of the scaled model, representing the collapse process during the test. The resultant load value refers to the cumulative horizontal force applied by all the hydraulic cylinders, whereas the displacement value is recorded by the transducer of D4. The recorded maximum force during the experiment is 1210 kN, followed by the failure of the test model with a significant increase in deformation. It can be seen that the funnel structure shows an almost linear behaviour before point

P1. It indicates that most members of the funnel are still within the elastic stage. When the external force increases about to 82 tons, intermittent noise occurs due to the plastic buckling of internal members. Before reaching the limit state point P2, it is obvious that the curve's slope begins to decline as the load increases. It means that the bending stiffness of the test model decreases due to the local buckling and failure of members. As the hydraulic loads are progressively increased, the weakest part experiences severe deformation leading to the detachment of multiple strain gauges. This signifies that the test model has reached its ultimate condition.

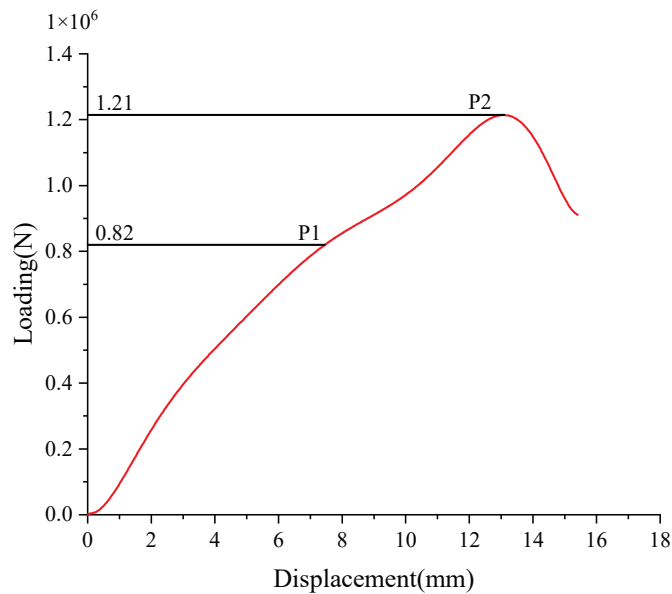
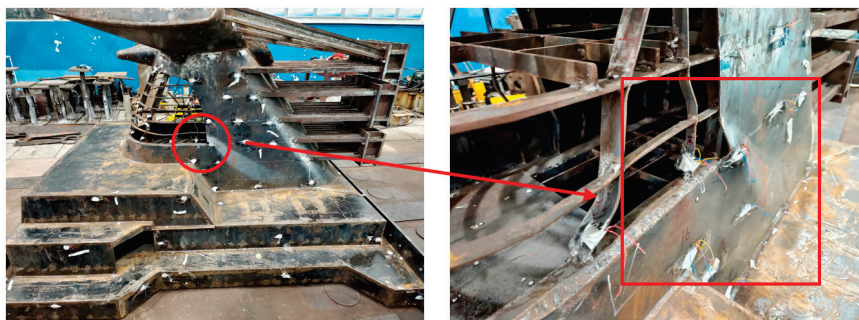


Figure 20. Load–displacement curve from the model test.

3.2. Collapse Mode

The specimen undergoes structural failure when subjected to a quick and substantial deformation, resulting in a considerable decrease in hydraulic forces. The large plastic deformation of local structures is shown in Figure 21. The sudden collapse of the funnel specimen is mostly due to plastic buckling occurring in its most vulnerable section. Under the action of horizontal wind pressure, the bottom section of the funnel body suffers the greatest bending moment, similar to that of a cantilever beam.



(a) Funnel plate

Figure 21. Cont.

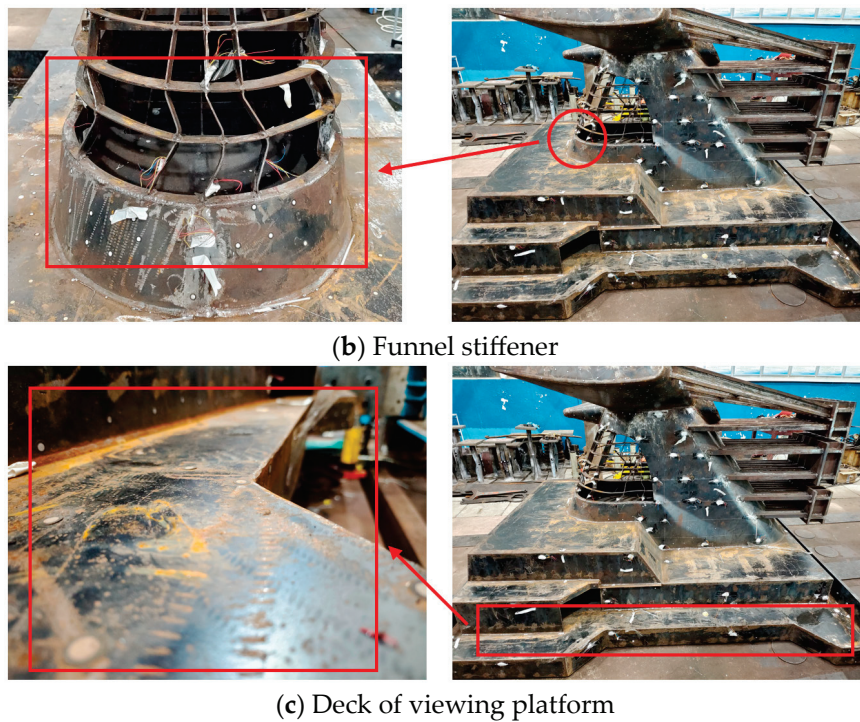


Figure 21. The deformation of the model.

The primary occurrence of plastic failure in the funnel panel is concentrated in the opening corner. The local plates have large deflection when buckling happens. The propagation of buckling rapidly extends to the nearby panels, resulting in the tripping of stiffeners. Finally, the structure experiences plastic failure in the opening section (Figure 21a). The stiffeners within the opening exhibit significant buckling distortion and collapse at the bottom end (Figure 21b). The decks of the viewing platform supporting the funnel body do not show large deformations (Figure 21c). It is reasonable because the decks suffer out-of-plane stresses during the collapse process and the deck panels are hard to have buckling other than the ones of the funnel body suffering compressive stresses.

3.3. Stress Distribution

The von-Mises stress can be derived from the data collected by the strain gauges. The obtained results for partial yield region measured points are shown in Figure 22a,b, corresponding to the gauge arrangement in previous figures. The material yield stress is taken as the maximum value when the calculated stress is above this threshold, as the formulas rely on the linear elastic theory. It is evident that the stress initially rises at Point 5, located at the upper edge of the opening, due to the relatively low stiffness of the local structure. Point 12 exhibits the largest stress and experience buckling first, which is located at the opening corner, followed by large plastic deformation. The primary concentration of stresses occurs in the buckling zones (Points 12 and 16), with significantly higher strains present at the vertical stiffeners within the opening (S11 and S6). The stresses (Points 12 and 16) are mainly concentrated around the buckling regions, and much larger stresses exist at the vertical stiffeners in the opening (S11 and S6). A noticeable characteristic of sudden stress increase is detected at several points (Points 20 and S1). This abrupt shift in stresses and strains is caused by the buckling deformation of certain components, implying that the specimen suffers plastic buckling failure.

The strain gauge results are symmetrical with respect to the longitudinal direction of the scaled model (Points 3 and 45, Points 11 and 48, Points 33 and 52, S7 and S12 and S6 and S15; see Figures 17 and 18), as shown in Figure 23. It is evident that the data from corresponding strain gauges are consistent throughout the testing process, suggesting that

the applied load and deformation exhibit a satisfactory left–right symmetry, matching the criteria for a collapse test.

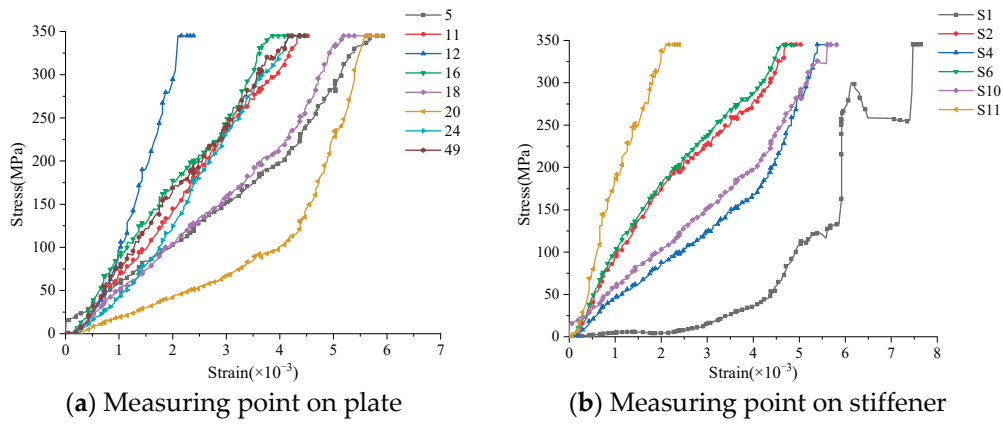


Figure 22. Average stress–strain relationship of the model.

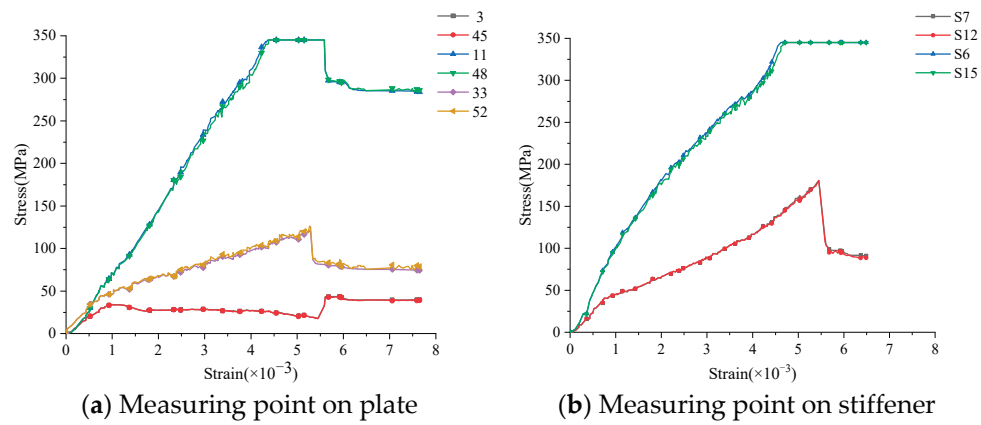


Figure 23. Test model symmetry check.

The strain gauge measurements taken from specific points of the deck (Points 36, 38, 54 and 55, as shown in Figure 17) are displayed in Figure 24. The obtained results from these points exhibit a linear change, indicating that the deck area of the test model is still in the elastic range, which corresponds to the previous results.

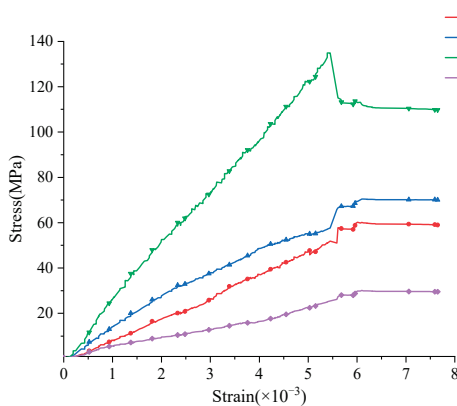


Figure 24. Measuring point on deck.

4. Comparison of the Experimental and Numerical Results

Experimental verification can be used to guarantee the precision of numerical computation outcomes. Firstly, the numerical load–displacement curve of the scaled model is

compared to the experimental result in Figure 25. The ultimate strength value analyzed from the numerical simulation is 1.24×10^6 N, while it is 1.21×10^6 N in the experiment, with a difference of 2.4%. Within the model's initial linear range, the two curves exhibit a strong concurrence. During the nonlinear stage, there is little discrepancy in predicting the ultimate strength, and the displacement obtained by the test is larger than that by the numerical analysis, with an error of 7.7%. At the beginning of the experiment, the funnel specimen undergoes multiple cycles of preloading and unloading processes to release the welding residual stress as well as reduce the gaps between the specimen and the supporting beam. However, the external force in the load–unload stage is not strong enough to eliminate the gaps in the specimen completely. The zero-displacement condition cannot be fully achieved in the experimental tests, which are applied in numerical analysis. In addition, some plates and stiffeners buckle seriously when subjected to loads closing to the limit state point. The test model experiences a sudden and significant deformation, leading to extensive destruction of the scaled model in a short period of time, which complicates the process of collecting data. Moreover, since the initial deflection and the residual welding stress cannot be recorded completely in the numerical analysis, there are discrepancies in the local buckling behaviour of some components. In any case, the finite element results are valid compared to the experimental ones with an acceptable margin of error. According to the similarity law, the ultimate strength value of the actual funnel can be predicted by using Equation (1), which is 3.01×10^7 N ($=1.21 \times 10^6 \times 15 \times 1.67$ N).

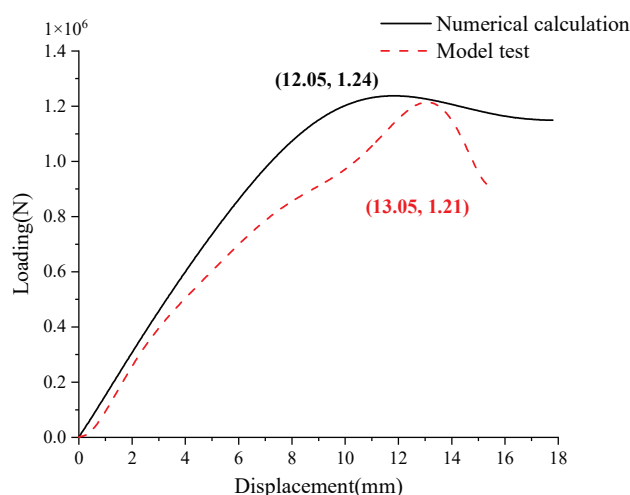


Figure 25. Comparison of load–displacement curves between numerical and experimental results.

5. Discussion

This discussion focuses on the specific characteristics that significantly impact the ultimate strength of funnel structures, such as the influence of the large opening, the primary constituting members, and local reinforcements. The instances and outcomes that were studied are listed in Table 4.

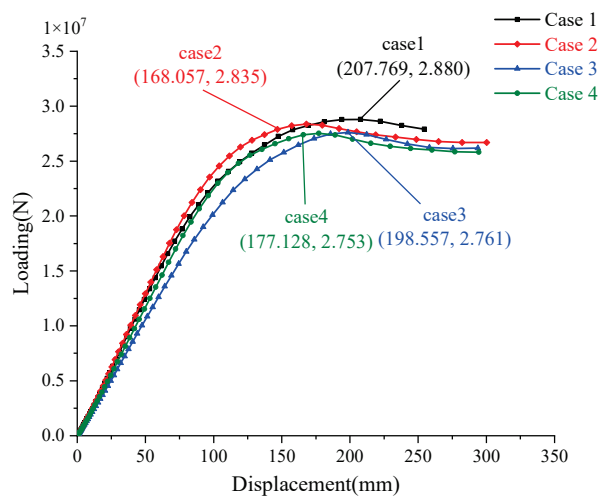
5.1. Effect of Reinforced Stiffeners

Additional finite element simulations are performed to examine the impact of the reinforcements surrounding the large opening on the ultimate strength of the funnel, as outlined in Cases 1–4 described in Table 4. A comparison of the load–displacement relationship is presented in Figure 26. By comparing Case 1 to Case 4, it shows that the ultimate strength is reduced by 4.4%. Upon analyzing the arrangement of reinforcement in the opening, it is found that the ultimate strength of the funnel with only vertical reinforcement decreases by 1.6% (Cases 1 and 2), while that with only transverse reinforcement decreases by 4.1% (Cases 1 and 3). It is analyzed that the plates and the stiffeners near the opening fail due to buckling under wind pressure. Thus, the vertical reinforcements can effectively improve the buckling strength other than the transverse ones.

Table 4. The lists of the selected scenarios.

Case	Reinforced Opening	Stiffener Thickness	Plate Thickness	Ultimate Strength
1	V + T	10 mm	5 mm	2.880×10^7 N
2	V	10 mm	5 mm	2.835×10^7 N
3	T	10 mm	5 mm	2.761×10^7 N
4		10 mm	5 mm	2.753×10^7 N
5	V + T	10 mm	7 mm	2.923×10^7 N
6	V + T	10 mm	9 mm	2.953×10^7 N
7	V + T	10 mm	11 mm	2.963×10^7 N
8	V + T	10 mm	13 mm	2.969×10^7 N
9	V + T	12 mm	5 mm	2.920×10^7 N
10	V + T	14 mm	5 mm	2.949×10^7 N
11	V + T	16 mm	5 mm	2.961×10^7 N
12	V + T	18 mm	5 mm	2.966×10^7 N
13	V + T	12 mm	7 mm	2.926×10^7 N
14	V + T	14 mm	9 mm	2.956×10^7 N
15	V + T	16 mm	11 mm	2.978×10^7 N
16	V + T	18 mm	13 mm	2.996×10^7 N

Note: V and T represent the vertical and transverse reinforcements, respectively.

**Figure 26.** Influence of reinforced opening.

5.2. Effect of Member Thickness

Figure 27a illustrates the correlation between the ultimate strength and the thickness of the member. The enhancement in the funnel's strength is somewhat more pronounced when increasing the plate thickness (Case M1) compared to increasing the reinforcement thickness (Case M2). When the increased thickness exceeds 4 mm, the growth rate of the ultimate strength of M1 and M2 decreases significantly. The ultimate strength of the funnel in case M3 does not increase largely compared to that in case M1 and M2 when the extra thickness is below 4 mm. However, the growth rate of the ultimate strength in case M3 has no remarkable decrease when the increased thickness exceeds 4 mm. The relationship between the increased ultimate strength and member mass is shown in Figure 27b. It is obvious that the increasing strength per unit weight in Case M1 is much larger than the other two cases. Therefore, it is better to improve the funnel's strength by applying larger plate thickness. As the mass grows, the expansion rate of the structural strength slows down in the cases of M1 and M2. It implies that several members should be simultaneously strengthened in order to compensate for this effect.

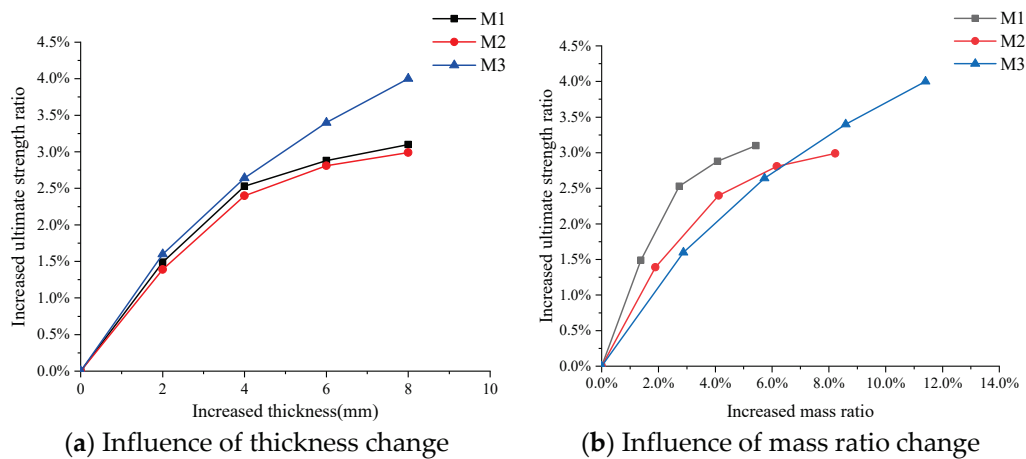


Figure 27. Influence of different factors on ultimate strength ratio. M1 represents thickening the plate only; M2 represents thickening the stiffener only; M3 represents thickening the plate and the stiffener at the same time.

5.3. Effect of the Large Opening

The ultimate strength of the funnel with no large aft opening is also analyzed using the FE method. Figure 28a shows the load–displacement curves, whereas Figure 28b illustrates the failure behaviour of the intact one. It can be seen that the opening reduces the ultimate strength by around 29.2%. When the opening is enclosed, the plastic buckling of the intact specimen primarily occurs at the joints between the top wings and the funnel body, marked in yellow lines. In order to improve the resistance of the funnel structure against buckling, it is essential to augment the rigidity of the open cross-section to match that of the closed cross-section.

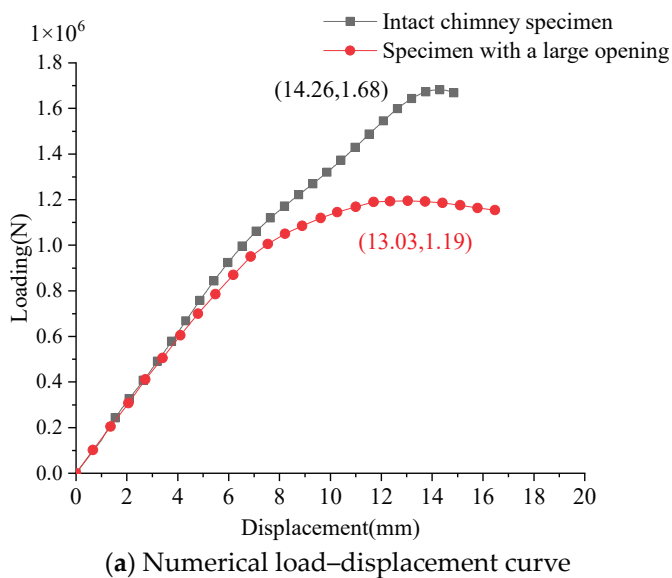


Figure 28. Cont.

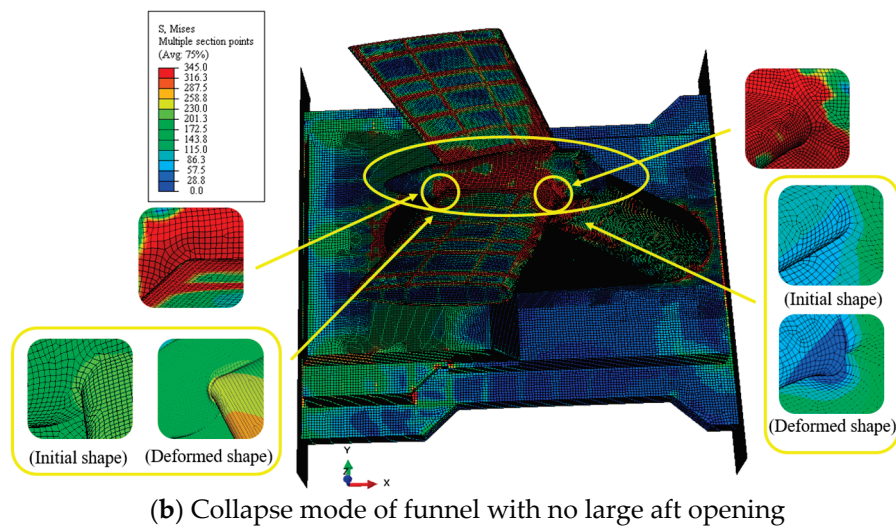


Figure 28. Influence of large opening.

6. Conclusions

In this study, both the experimental test and finite element calculation are conducted to assess the ultimate strength of a large passenger ship's funnel structure under wind load, which reveals the collapse pattern of the ship's funnel with openings. It introduces a comprehensive experiment for evaluating the ultimate strength of a similarly scaled funnel model under wind pressure. It illustrates the mechanical behaviour related to the ultimate strength and the step-by-step collapse of the ship funnel. Additionally, the corresponding numerical simulation is performed, and the numerical results match closely the experimental data in terms of the load–displacement curves, plate deformation and stresses. From the findings of the experimental and numerical analyzes, several key insights are concluded as follows:

- (1) The designed testing device is valid to apply large wind loads, and the scaled model can accurately present the progressive collapse process of the ship funnel subjected to wind pressure, with an error of 3.5% for predicting the ultimate strength value. The collapse patterns of structures observed in the experiment coincide with those derived from numerical analysis.
- (2) The presence of a large opening significantly alters the collapse pattern of the funnel structure, reducing its ultimate strength by 29.2%. The buckling deformation primarily occurs at the opening's corner, followed by the sudden collapse of the specimen due to plastic deformation at the opening cross-section. The funnel tends to lean backward like a cantilever beam under horizontal wind pressure. The plates and stiffeners near the opening are prone to buckling under compression, and the side plates facing the wind are subjected to tensile stress.
- (3) Reinforcing the members surrounding the aperture increases the bending stiffness of the opening section and finally enhances the funnel's ultimate strength. Thickening the plate material is found to be more effective than merely increasing the thickness of internal stiffeners. The optimal approach for improving structural strength involves ensuring the open section has comparable stiffness to the closed ones, thereby shifting the buckling region to the structures far from the opening. This principle is crucial for the optimal design of structures with inevitable openings.

At present, the research conducted in this paper is based on the equivalent static wind load, and it is evenly distributed on the windward side of the funnel. But besides the static component, the wind pressure also has a pulsation component. As the chimney has a complex shape, the distribution of wind pressure over its surface differs significantly from uniform. In the future, CFD software (ANSYS Fluent 2023 R1) complexes can be used to carry out these studies.

Author Contributions: Conceptualization, Z.D. and L.A.; methodology, Z.D.; software, Z.D.; validation, L.A.; writing—original draft preparation, Z.D.; writing—review and editing, L.A., B.L., Z.P., Q.T. and W.W.; supervision, L.A., B.L. and Z.P. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by The general project of the National Natural Science Foundation of China, grant number 52301385, The Fundamental Research Funds for the Central Universities, grant number WUT:223173003.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data supporting the reported results cannot be shared at this time, as it were used in producing more publications on this research.

Acknowledgments: This work was supported by Hubei Provincial Engineering Research Center on Green & Smart River-sea-going Ship, China.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Janssen, W.D.; Blocken, B.; van Wijhe, H.J. CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications. *J. Wind. Eng. Ind. Aerodyn.* **2017**, *166*, 106–116. [CrossRef]
2. Bahatmaka, A.; Kim, D.; Prabowo, A. Numerical Investigation against Laboratory Experiment: An Overview of Damage and Wind Loads on Structural Design. *Procedia Struct. Integr.* **2020**, *27*, 6–13. [CrossRef]
3. Chou, J.; Chiu, C.; Huang, I.; Chi, K.-N. Failure analysis of wind turbine blade under critical wind loads. *Eng. Fail. Anal.* **2013**, *27*, 99–118. [CrossRef]
4. Ishihara, T.; Yamaguchi, A.; Takahara, K.; Mekaru, T.; Matsuura, S. An analysis of damaged wind turbines by typhoon maemi in 2003. In Proceedings of the Sixth Asia-Pacific Conference on Wind Engineering, Seoul, Republic of Korea, 12–14 September 2005.
5. Bajic, D. Structural design of passenger cruise ships—An introduction to classification requirements. *Ships Offsh Struct.* **2015**, *10*, 232–238. [CrossRef]
6. Jebsen, J.J.; Papakonstantinou, V.C. Evaluation of the Physical Risk of Ship Grounding. Master’s Thesis, Massachusetts Institute of Technology, Cambridge, MA, USA, 1997.
7. Wang, P.; Wang, F.; Chen, Z. Investigation on aerodynamic performance of luxury cruise ship. *Ocean. Eng.* **2020**, *213*, 107790. [CrossRef]
8. Doan, V.T.; Liu, B.; Garbatov, Y.; Wu, W.; Guedes Soares, C. Strength assessment of aluminium and steel stiffened panels with openings on longitudinal girders. *Ocean. Eng.* **2020**, *200*, 107047. [CrossRef]
9. Zhou, B.; Han, X.; Guo, W.; Liu, Y.; Tan, S.K. Numerical and experimental study on cutting access opening in ship structure. *J. Ship Prod. Des.* **2017**, *33*, 12–23. [CrossRef]
10. Zhao, Z.; Zhang, M.; Gao, Y.; Sun, Q. Investigations on shear capacity of steel plates with local opening. *J. Constr. Steel Res.* **2021**, *179*, 106518. [CrossRef]
11. Liu, B.; Du, X.K.; Gan, J.; Ao, L.; Wu, W.; Soares, C.G. Experimental and numerical analysis of the ultimate compressive strength of double-deck structures with a large opening. *Ships Offshore Struct.* **2022**, *17*, 2788–2801. [CrossRef]
12. Liu, B.; Gao, L.J.; Ao, L.; Wu, W. Experimental and numerical analysis of ultimate compressive strength of stiffened panel with openings. *Ocean. Eng.* **2021**, *220*, 108453. [CrossRef]
13. Morshedsolouk, F.; Khedmati, M.R. An extension of coupled beam method and its application to study ship’s hull-superstructure interaction problems. *Lat Am J Solids Struct.* **2011**, *8*, 265–290. [CrossRef]
14. Romanoff, J.; Remes, H.; Varsta, P.; Jelovica, J.; Klanac, A.; Niemelä, A.; Bralic, S.; Naar, H. Hull-superstructure interaction in optimised passenger ships. *Ships Offshore Struct.* **2012**, *8*, 612–620. [CrossRef]
15. Yao, T.; Fujikubo, M.; Yanagihara, D.; Fujii, I.; Matsui, R.; Furui, N.; Kuwamura, Y. Buckling collapse strength of ship carrier under longitudinal bending (1st Report). *J. Soc. Nav. Archit. Jpn.* **2002**, *191*, 255–264. [CrossRef]
16. Benson, S.; AbuBakar, A.; Dow, R.S. A comparison of computational methods to predict the progressive collapse behaviour of a damaged box girder. *Eng. Struct.* **2013**, *48*, 266–280. [CrossRef]
17. Hirdaris, S.E.; Bai, W.; Dessi, D.; Ergin, A.; Gu, X.; Hermundstad, O.A.; Huijsmans, R.; Iijima, K.; Nielsen, U.; Parunov, J.; et al. Loads for use in the design of ships and offshore structures. *Ocean. Eng.* **2014**, *78*, 131–174. [CrossRef]
18. Quispe, J.P.; Estefen, S.F.; de Souza, M.I.L.; Chujutalli, J.H.; Amante, D.D.A.M.; Gurova, T. Numerical and experimental analyses of ultimate longitudinal strength of a small-scale hull box girder. *Mar. Struct.* **2022**, *85*, 103273. [CrossRef]
19. Pei, Z.Y.; Yuan, Q.N.; Ao, L.; Wu, W. A research on similarity law for combined ultimate bending and torsional strength. *Ships Offshore Struct.* **2023**, 1–12. [CrossRef]
20. Kulkarni, P.R.; Singh, S.N.; Seshadri, V. Parametric Studies of Exhaust Smoke-superstructure Interaction on a Naval Ship Using CFD. *Comput Fluids.* **2007**, *36*, 794–816. [CrossRef]

21. Chen, Q.; Yin, Q.; Fan, A.; Sun, X.; Mou, X. Research on the calculation methods of wind load coefficients of inland cruise ship. In Proceedings of the 2015 International Conference on Transportation Information and Safety (ICTIS), Wuhan, China, 25–28 June 2015; pp. 871–876.
22. Zhang, J.F.; Pei, H.; Li, J.; Ge, Y.J.; Zhao, L. Failure process and ultimate strength of RC hyperbolic cooling towers under equivalent static wind loads. *Thin-Walled Struct.* **2022**, *176*, 109307. [CrossRef]
23. Bhatt, M.; Vasanwala, S.A. Ultimate Strength of RC Chimney Section under Monotonic Loading. *J. Eng. Proj. Prod. Manag.* **2022**, *12*, 217–223.
24. Wang, C.; Wu, J.; Wang, D. Experimental and numerical investigations on the ultimate longitudinal strength of an ultra large container ship. *Ocean. Eng.* **2019**, *192*, 106546. [CrossRef]
25. Zhang, S.; Khan, I. Buckling and ultimate capability of plates and stiffened panels in axial compression. *Mar. Struct.* **2009**, *22*, 791–808. [CrossRef]
26. Dassault, S. *Abaqus Analysis User's Guide*; Dassault Systèmes Simulia Corp.: Providence, RI, USA, 2014.
27. Wang, Q.; Wang, C.; Wu, J.; Wang, D. Investigations on the torsional failure characteristics of the global hull girder with large deck openings. *Ocean Eng.* **2020**, *198*, 107007. [CrossRef]
28. Jimenez-Martinez, M.; Varela-Soriano, J.; Carreón, J.; Torres-Cedillo, S.G. Waveform load analysis for fatigue in the printed PLA. *Heliyon* **2023**, *9*, e18480. [CrossRef]
29. Hoang, T.; Duhamel, D.; Foret, G. Wave finite element method for waveguides and periodic structures subjected to arbitrary loads. *Finite Elem. Anal. Des.* **2020**, *179*, 103437. [CrossRef]
30. Liang, Y.; Ren, X. Failure analysis and curvature-based failure criterion of super large cooling tower under strong equivalent static wind load. *Eng. Fail. Anal.* **2022**, *142*, 106801. [CrossRef]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

XFEM-Based Study of Fatigue Crack Propagation in Rocket Deflector Troughs under Coupled High-Temperature and Impact Conditions

Zhixin Xiong ^{1,*}, Chengyuan Zhu ², Yue Yang ³, Tong Lin ³ and Ruoxuan Li ¹

¹ College of Ocean Science and Engineering, Shanghai Maritime University, Shanghai 201306, China

² Merchant Marine College, Shanghai Maritime University, Shanghai 201306, China

³ Marine Design & Research Institute of China, Shanghai 200011, China

* Correspondence: zxxiong@shmtu.edu.cn

Abstract: This research investigated fatigue crack propagation on the lower surface of rocket deflector troughs in offshore rocket launch platforms. Initially, a numerical model of an offshore rocket launch platform was established using ABAQUS based on the Extended Finite Element Method (XFEM). Subsequently, two variable parameters—the initial crack length and initial tilt angle—were introduced. This research systematically analysed the impact of these parameters on the fatigue crack propagation patterns in both the maximum stress and maximum deformation regions of the deflector channels under the combined conditions of high temperature and impact. Finally, the research indicated that the propagation length of surface cracks in the deflector trough exhibited an initial increase followed by a decrease with the rise in the pre-set inclination angle. Notably, the stable propagation rate of the crack in the region of maximum deformation surpassed that observed in the region of maximum stress. Through meticulous comparative analysis, it was evident that temperature loading significantly exacerbated the initiation and propagation of cracks, particularly in the upper region of the deflector channel's lower surface.

Keywords: offshore rocket deflector troughs; fatigue crack propagation; XFEM; high temperature and impact coupling

1. Introduction

As the frequency of space launch missions continues to rise, the deployment of rocket deflector troughs is increasing correspondingly, introducing inherent structural fatigue challenges. According to the statistics, more than 70% of the damage to marine engineering structures during their service life is caused by fatigue [1]. Consequently, the initiation and propagation of cracks due to structural fatigue have emerged as the primary causes of marine engineering structure failures. In the context of the challenging maintenance conditions of offshore rocket launch platforms, it is imperative to comprehend the trajectory from crack initiation to structural failure within the direct impact zone of rocket plumes on deflectors. This understanding holds significant importance for evaluating the remaining structural life of these platforms, particularly in light of stringent damage tolerance requirements.

The Extended Finite Element Method (XFEM), as an emerging numerical approach for simulating fatigue crack propagation, retains the advantages of traditional finite element methods. It utilises specialised crack tip enrichment functions to model the stress singularity at the crack tip, allowing the crack interface to be independent of the mesh, reducing mesh dependence, and simplifying the treatment of the crack tip refinement [2]. This makes the XFEM an effective and efficient tool for addressing crack growth problems [3–5]. Numerous scholars have conducted in-depth research and experimental extensions on this method. Patzak et al. [6] conducted an in-depth study on the analytical process of the XFEM. Zhang et al. [7] used the Extended Finite Element Method to study the impact of compressive

stress on crack propagation under tension-compression loading, revealing a relationship between crack initiation at the crack tip and compressive stress. Menouillard et al. [8] simulated dynamic crack propagation using the XFEM based on explicit time integration techniques. Fries et al. [9] introduced three horizontally integrated functions explicitly computed to enhance the three-dimensional applicability of the XFEM, proposing a new approach that combines explicit and implicit methods for simulating crack propagation. Daux et al. [10] applied the Extended Finite Element Method to simulate crack branching. Himanshu Pathak et al. [11] used the XFEM to simulate fatigue crack propagation under cyclic thermal loads for three-dimensional linear elastic cracks with different geometries, revealing the robustness and versatility of the XFEM. Loehnert et al. [12] discussed a series of computational issues related to the three-dimensional XFEM, studying the impact of finite deformation theory on crack simulation through numerical examples. Danas et al. [13] used the XFEM to study energy minimisation in the multiple crack propagation of linear elastic solids under quasi-static conditions, proposing an alternative strategy to supplement the implementation and application of gradient descent methods within the XFEM framework.

While conducting in-depth research on the XFEM, many scholars have also applied this method to address practical problems. Through experiments applying compressive stress on notched plates, Fleck et al. [14] found that cracks originated in the residual tensile stress region near the notch root and extended, indicating that fatigue cracks are more prone to propagate under tensile stress conditions. Sahouryeh et al. [15], through experiments, observed the crack propagation behaviour under biaxial compressive stress and found that the size of crack growth was limited, with the propagation direction parallel to the loading direction. Nikfam et al. [16] simulated high-cycle fatigue tests on welded T-joints using the XFEM, validating that the XFEM predictions of crack propagation rate, life, and shape were within a reasonable range of the experimental results, with the fracture morphology of the numerical model consistent with experimental observations. Teng et al. [17] utilised the XFEM to investigate the influence of different micro-defects on macro-crack propagation. The study results not only demonstrated the XFEM's ability to accurately capture the stress intensity factors at the macro-crack and micro-defect crack tips but also revealed that the impact of micro-defects on the macro-crack propagation path changed with their position and size.

Based on simulated rocket plume parameters, this study employed numerical simulation to investigate the influence of the rocket plume on the launch platform, especially the deflector, during rocket launch. The simulation incorporated both high-temperature and exhaust plume impact load coupling, considering the effects of temperature loads on the mechanical properties of the deflector's metallic material. The force conditions on the deflector during the rocket's takeoff phase were realistically simulated. Finally, the XFEM was applied to analyse the fatigue crack propagation on the lower surface of the rocket deflector with pre-existing initial cracks. This study primarily focuses on the direction, initiation length, and fatigue propagation rate of cracks in rocket deflectors with initial cracks caused by fatigue damage or material defects during their subsequent operational service. The obtained results will serve as a reference for the safety performance design of maritime rocket deflectors.

2. Models and Environmental Parameters

2.1. Rocket Launch Platform Model

The research in this paper focuses on the lower surface of a rocket deflector trough subjected to direct impingement from the rocket plume, as illustrated in the overall model of an offshore rocket launch platform shown in Figure 1a.

Given that the high-temperature and high-speed airflow during rocket launches primarily affects the launch area of the platform (the main body of the deflector trough and the connected upper deck region), and considering the symmetrical distribution of the deflector trough, the right side of the deflector trough was selected as the representation

area for computational results to simplify the visualisation, as illustrated in Figure 1b. During a rocket launch, the plume jet traverses the deflector trough inlet positioned at the centre of the launch deck, exerting a direct impact on the deflector trough beneath the deflection hole. Subsequently, it follows the guidance of the deflector's lower surface, ultimately exiting from the openings at both ends of the deflector. The dimensions of the deflector trough are illustrated in Figure 2.

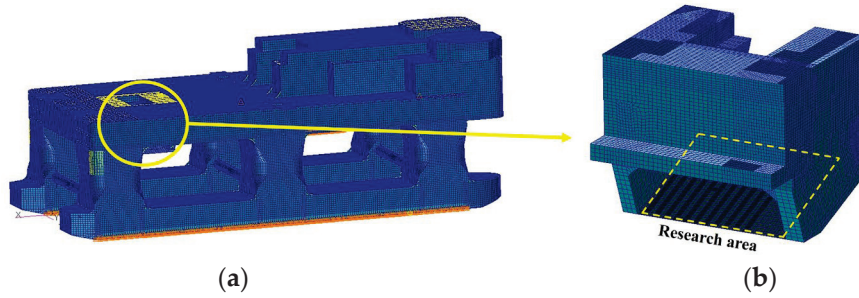


Figure 1. Schematic diagram of offshore rocket launch platform (a) and deflector trough model (b).

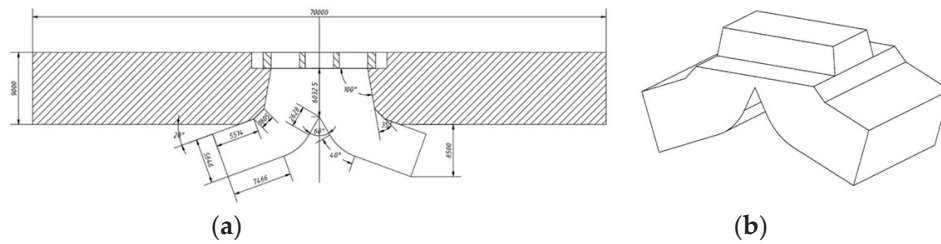


Figure 2. Diagram of deflector dimensions; (a) Overall dimensions diagram of the rocket deflector; (b) Overall profile diagram of the rocket deflector.

2.2. Environmental Load Parameters

The external environmental parameters of the deflector are primarily divided into temperature parameters and exhaust plume impact load parameters. The conventional method to ensure the structural safety of the rocket deflector is to install heat insulation materials on the surface of the deflector, which can significantly reduce the temperature of the metal structure directly exposed to the rocket plume. By incorporating sensors beneath the insulating layer on the lower surface of the deflector, it becomes possible to capture data regarding temperature variations and exhaust plume impact loads during the initial 1.1 s of a rocket launch. Figures 3 and 4 illustrate the numerical values of temperature and impact force at specific moments, visually representing these critical parameters.

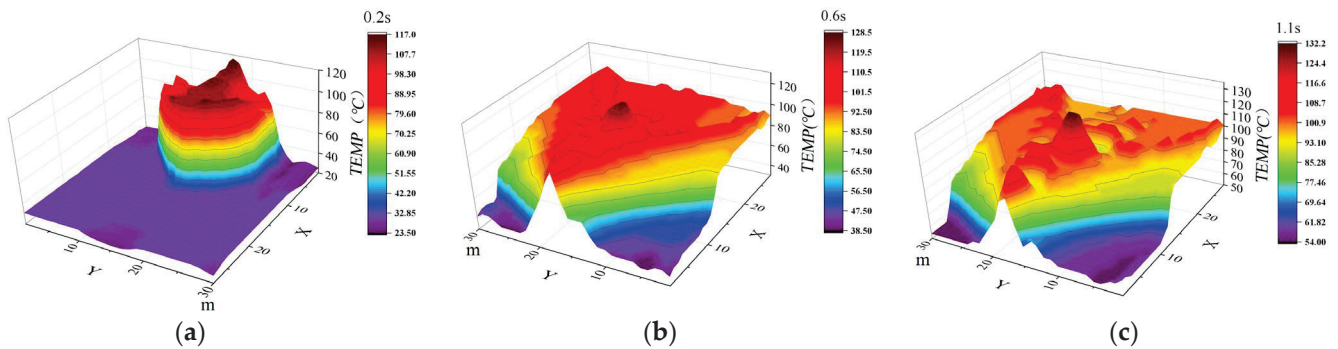


Figure 3. Temperature contour plot: (a) 0.2 s, (b) 0.6 s, (c) 1.1 s.

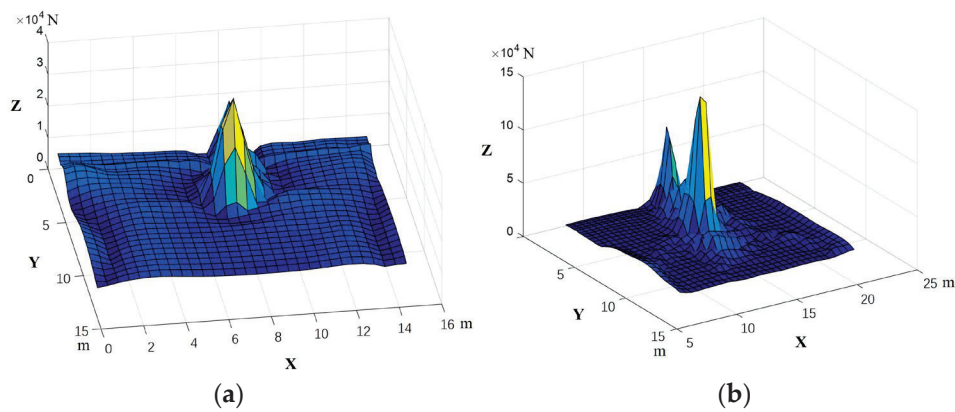


Figure 4. Impact force load cloud map; (a) 0.2 s; (b) 1.1 s.

The boundary conditions for the overall model were defined as an air environment with a pressure of 101,325 Pa and an ambient temperature of 20 °C. There was no heat exchange between the deflector and the external environment. The deflector walls were treated as no-slip surfaces. The boundary constraints were applied to both the symmetry planes of the deflector and the region where the deflector is connected to the upper deck of the platform, involving fixed displacement and rotation. The material chosen for the rocket deflector was grade E ordinary shipbuilding steel. The chemical composition of this steel is: C ≤ 0.17%, Mn ≤ 1.4%, Si ≤ 0.35%, P ≤ 0.035%, S ≤ 0.035%, Al ≥ 0.015%. It is crucial to note that the increase in temperature not only diminishes the elastic modulus of the metal material but also attenuates its yield strength [18]. Consequently, both these material properties are dynamically adjusted with temperature variations [19], as comprehensively outlined in Table 1 [20]. The temperature referenced for the fracture mechanics properties of the deflector metal material is above 200 °C. In comparison, this study's maximum temperature on the lower surface was 132.2 °C. The actual temperature of the metal is lower than this, because this study measured the temperature on the backside of the insulation material. The fracture mechanics properties of the deflector material were set using room temperature parameters, as detailed in Table 2 [21].

Table 1. Elastic modulus and yield strength of grade E shipbuilding steel at various temperatures.

Temperature (°C)	Elastic Modulus (10 ⁵ MPa)	Yield Strength (MPa)
16	2.032	299.0
200	1.950	245.8
250	1.930	229.0

Table 2. The fracture mechanics properties of grade E steel at room temperature (20 °C).

G_{IC} (N/mm)	C	m	ΔK_{th} (N/mm ^{3/2})
80.0330	2.2644×10^{-16}	4.1987	225.7000

The material parameters c_1 and c_2 were used to judge the crack from absence to presence and were taken as 0.5 and −1, respectively. The parameters c_3 and c_4 were related to the material's intrinsic properties and were taken as 3.74×10^{-5} and 2.0994, respectively.

3. Simulation Calculation Equations

3.1. XFEM

The fundamental idea of the XFEM [22] is the method of unit decomposition, which introduces enriched functions into the displacement modes of FEM to characterise strong discontinuities in the displacement field:

$$u^h = \sum_i N_i(x) u_i + \varphi(x) \quad (1)$$

where u^h is the discontinuous displacement field; standard finite elements apply the shape function; u_i represents the degrees of freedom of regular nodes; and $\varphi(x)$ represents the enriched approximation functions associated with the method of unit decomposition.

When applying the XFEM to solve crack propagation, the software divides the material elements into three categories: standard elements, penetrating elements, and crack tip elements. For the elements experiencing crack propagation, discontinuous shape functions can be introduced to control the displacement:

$$\begin{cases} u_{xfem}(x) = u_{fem}(x) + u_{step}(x) + u_{tip}(x) \\ u_{fem}(x) = \sum_{i \in I} N_i(x) u_i \\ u_{step}(x) = \sum_{i \in I_{step}} N_i(x) H(x) a_i \\ u_{tip}(x) = \sum_{i \in I_{tip}} N_i(x) F(x) b_i \end{cases} \quad (2)$$

In the equation above, $u_{xfem}(x)$ represents the approximate displacement field of the extended finite element; $u_{fem}(x)$ represents the approximate displacement field of the standard finite element; $u_{step}(x)$ represents the approximate displacement field of the penetrating element crack; $u_{tip}(x)$ represents the approximate displacement field of the crack in the crack tip element; I is the set of all nodes in the solution domain; I_{step} is the set of nodes penetrated by the crack; I_{tip} is the set of nodes affected by the crack tip region; $H(x)$ is the step function for the penetrating element; $F(x)$ is the enrichment function for the crack tip element; a_i represents the degrees of freedom associated with enriched nodes related to the jump discontinuity; and b_i represents the degrees of freedom associated with enriched nodes related to the crack tip approximation function.

3.2. Crack Propagation Criteria

The Paris equation can represent the initiation and propagation of fatigue cracks [23]:

$$\frac{da}{dN} = c_3 (\Delta G)^{c_4} \quad (3)$$

In the equation above, c_4 represents material-related constants.

The Paris law in the simulation associates the crack propagation rate with the energy release rate (ΔG) at the crack tip. $\Delta G = G_{\max} - G_{\min}$, $G_{\max}$ represents the strain energy release rate corresponding to the maximum load on the structure and the strain energy release rate corresponding to the minimum load on the structure.

The Paris law operates within the bounds of G_{thresh} and G_{pl} , where fatigue cracks do not propagate below the former and rapidly propagate until failure above the latter. The intermediate region between the two represents a stable crack propagation zone for fatigue cracks.

3.3. Damage Initiation Criteria and Evolution Criteria in ABAQUS

(1) Damage Initiation Criterion: Maximum Principal Stress Criterion (Maxps Damage)

$$f = \left\{ \frac{\langle \sigma_{\max} \rangle}{\sigma_{\max}^0} \right\} \quad (4)$$

When $1.0 < f < 1.0 + f_1$ within the tolerance range, damage initiates. Here, f_1 is the tolerance, typically set to 0.05; $\sigma_{\max}^0$ is the critical maximum principal stress value; and $\langle \sigma_{\max} \rangle$ signifies

$$\begin{cases} \langle \sigma_{\max} \rangle = 0, \sigma_{\max} < 0 \\ \langle \sigma_{\max} \rangle = \sigma_{\max}, \sigma_{\max} \geq 0 \end{cases} \quad (5)$$

The above expression implies that no damage initiation occurs in the element with a penetrating crack under pure compressive loading conditions.

(2) Damage Evolution Criterion: Energy-Based Damage Evolution Criterion

In ABAQUS, the overall damage of a structure is represented by D , with an initial value of 0. If a damage evolution model is defined, the value of D increases after the model undergoes further loading, gradually evolving towards 1. ABAQUS's damage evolution law describes the degradation rate of material stiffness in each cycle once the corresponding initiation criteria are met. For ductile materials, ABAQUS assumes that stiffness degradation can be modelled using a scalar damage variable D . Within any given analysis cycle, the stress tensor in the material is determined by the scalar damage equation:

$$\sigma = (1 - D)\bar{\sigma} \quad (6)$$

The equation σ represents the effective (undamaged) stress tensor present in the material during the current incremental calculation when no damage has occurred. When D equals 1, it indicates that the material has lost its load-bearing capacity, and damage has occurred [24].

(1) Linear Evolution Damage Criterion:

$$D = \frac{s_m(s_{\max} - s_0)}{s_{\max}(s_m - s_0)} \quad (7)$$

In the equation above, $s_m = \frac{2G}{F}$, F is the effective traction measured in MPa, and G is the energy released upon damage measured in J.

(2) Exponential Evolution Damage Criterion:

$$D = \int_{s_0}^{s_m} \frac{F d_s}{G - G_0} \quad (8)$$

In the equation above, G_0 is the energy release at the onset of damage measured in J, and s is the effective displacement measured in mm.

4. Experimental Validation of the Effectiveness of the Simulation Method

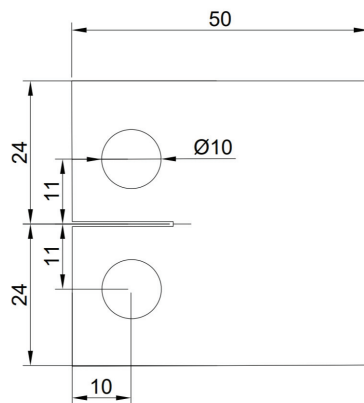
4.1. Experimental Setup

The material used for the fatigue crack propagation experiment was the GH4133B alloy, in the form of standard CT specimens. The stress ratio for the fatigue experiment was set to 0.02 ($R = 0.02$), and a sinusoidal wave with a frequency of 5 Hz was employed as the experimental loading frequency. The tensile stress direction was set as symmetrically reversed [25].

The measurement method for fatigue crack propagation was as follows: After subjecting the specimen to a certain number of cyclic stress loadings, the test machine was temporarily halted. Subsequently, the specimen's crack was examined utilising an electron microscope, with images of the crack tip captured. These images were then juxtaposed with prior observation results to trace the path of fatigue crack propagation, concurrently documenting the associated cycle counts. The specimen parameters are shown in Table 3 and Figure 5, and the tested specimen is shown in Figure 6.

Table 3. Specimen parameters.

Specimen Number	Symmetric Load (P/kN)	Specimen Thickness (W/mm)
1	7	6.8
2	6.5	6.8
3	8.5	6.4
4	8.8	6.8

**Figure 5.** Specimen dimensions and loading configuration.**Figure 6.** Experimental specimen.

4.2. Comparison between Experimental Data and Simulation Data

The stress cycle counts obtained through experimental measurements and simulated by the Extended Finite Element Method when the crack propagated to a length of 15 mm are presented in Table 4. The relationship between the crack length (a) and the number of cycles (N) for specimens measured through experimental methods and simulated using the XFEM is illustrated in Figure 7.

Table 4. Comparison of experimental and simulation results.

Specimen Number	Number of Cycles in the Experimental Method (N/Cycle)	Number of Cycles in the Simulation Method (N/Cycle)	Relative Error (%)
1	135,375	140,515	3.80
2	179,672	175,050	2.57
3	50,080	50,975	1.79
4	70,806	71,575	1.09

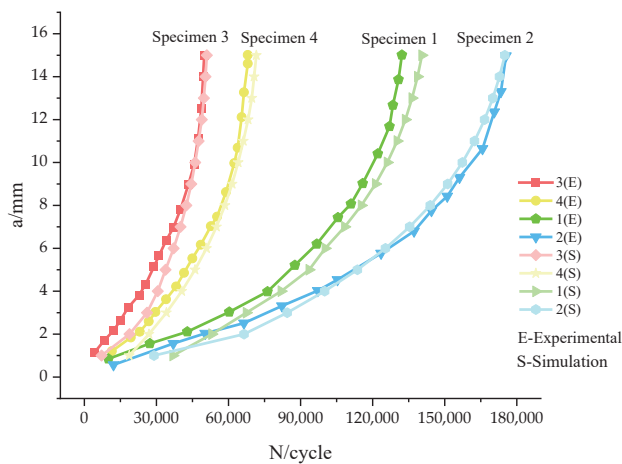


Figure 7. Comparison of experimental and simulation results.

Reviewing Table 4, it becomes apparent that the maximum error in the number of cycles measured by the two methods was 3.8%. Figure 7 illustrates that the crack propagation patterns of the specimens obtained through both methods generally aligned under equivalent thickness and load conditions. The observed trend indicated an increment in crack propagation rates with the number of load cycles, affirming the reliability of XFEM simulation.

5. Numerical Results and Discussion

5.1. XFEM Simulation Approach for Fatigue Crack Growth

This paper employed the XFEM to simulate the numerical growth process of cracks. The XFEM, by introducing additional shape functions into the finite element mesh, can handle the extension of cracks more naturally. In the calculation of crack propagation, the XFEM employs the concept of unit decomposition to introduce specific functions relevant to the problem into the approximation space, ensuring the convergence of results [26]. The level set method is a numerical technique within the XFEM used to determine the position of the internal interface of cracks and to track the growth of cracks. Any internal interface can be represented using its zero-level set function [27,28].

In the simulation, the release of nodes at the crack tip was based on ABAQUS damage initiation and evolution criteria. The fracture energy in the crack tip region accumulated as the load cycled continuously. When the cumulative fracture energy at the crack tip nodes reached a specific threshold, the XFEM released the fracture energy at the tip, and this released energy was used to initiate a new crack, simulating the extension of the crack. ABAQUS abandons the coupling issues introduced by two shape functions, making it unable to simulate branching cracks. However, the XFEM, utilising the unit decomposition and level set methods, can accurately simulate the extension of a single crack. Therefore, in this paper, crack propagation refers to the growth and elongation of cracks caused by cyclic loading and does not include an increase in the number of cracks.

The growth of the crack in each numerical step was determined by monitoring the stress intensity factor range at the crack tip [29]. Using empirical relationships like the Paris law, the crack growth rate was associated with changes in the stress intensity factor, allowing for the crack growth calculation in each step. This ensured that the XFEM can simulate the actual growth behaviour of cracks under fatigue loading.

5.2. Crack Area Configuration

The crack locations were chosen based on the stress–strain calculation results of the deflector’s lower surface. Two positions were selected, one corresponding to the maximum stress and the other to the maximum strain. Figure 8 indicates the specific locations of the cracks along with the results of stress/strain calculations. Position A corresponds to the location with the maximum stress, as shown in part (a). Position B corresponds to the

location with the maximum strain, as indicated in part (b). The region circled in yellow represents the area designated for pre-inserted cracks. The width of the lower surface of the deflector plate was 20 m. According to the design requirements for the launch platform, the crack size could not exceed 5‰ of the deflector plate width. This paper conducted simulations under the worst-case-scenario assumption; hence, the initial crack length was set to 100 mm.

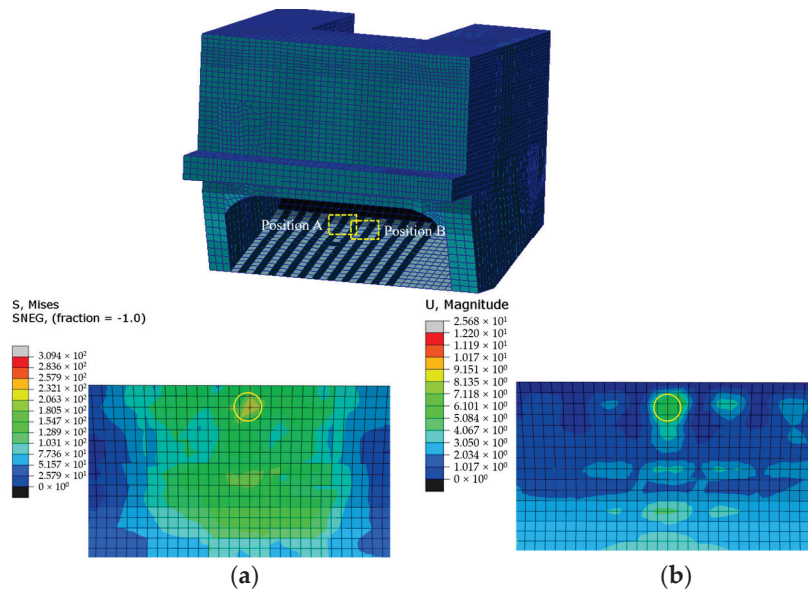


Figure 8. Schematic diagram of operating condition zones; (a) Condition A: Cracks are positioned at the locations of maximum stress; (b) Condition B: Cracks are positioned at the locations of maximum deformation.

Considering that the crack propagation on the lower surface of the deflector occurred only in localised regions, a “shell-to-solid” coupling theory based on ABAQUS [30] was adopted. Figure 9 illustrates the shell–solid coupling principle. In this approach, the localised regions where cracks must pre-exist are configured as three-dimensional solids. In contrast, the overall model region is kept as a two-dimensional shell. This configuration was implemented to simplify the computational process. The form of the crack was characterised as a penetrating crack.

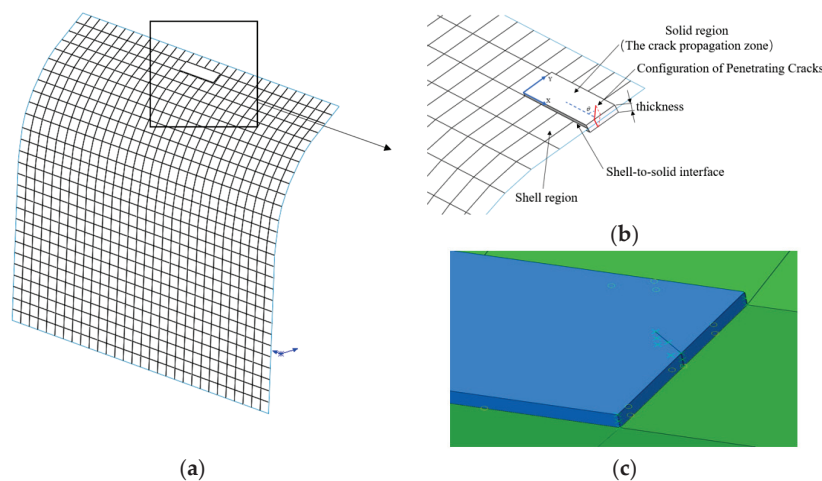


Figure 9. Shell-to-solid coupling schematic diagram; (a) Overall; (b) Local detailed illustration; (c) Crack configuration.

The cracks studied in this paper were penetrating cracks that vertically traversed the lower plate of the deflector groove. The plane coordinate system selected for measuring the initial crack's inclination angle was located on the plane in the region where the crack was present. Figure 9b,c illustrates the configuration of penetrating cracks and the establishment of the coordinate system referenced by the inclination angle of the crack.

5.3. Crack Propagation Direction

The pre-inserted cracks on the lower surface of the rocket deflector experienced various types of stress, including tensile stress, compressive stress, torsional stress, and shear stress. Due to the non-uniformity in the motion of the rocket plume flow field, there were variations in the temperature and exhaust plume impact loads, both in terms of direction and magnitude, across different regions of the lower surface of the deflector trough. As a result, cracks at different positions and inclinations experienced differences in the magnitude and direction of the principal stress. Figures 10 and 11 depict schematic diagrams illustrating the inclination and propagation of pre-inserted cracks under the propagation scenario for two operational conditions, both occurring within the same number of load cycles. In the figures, the X-axis direction corresponds to 0° , and the Y-axis direction to 90° . The counterclockwise direction represents an increase in inclination. For clarity, the crack represented by the thick line in the figure is the initially pre-set crack in the deflector groove. In contrast, the crack represented by the thin line is the newly formed portion resulting from the fatigue-induced crack propagation of the pre-set initial crack.

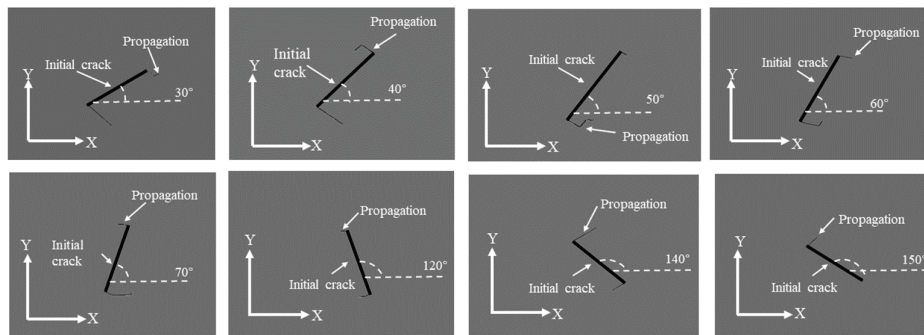


Figure 10. The crack propagation results under different pre-set inclinations for operational condition A.

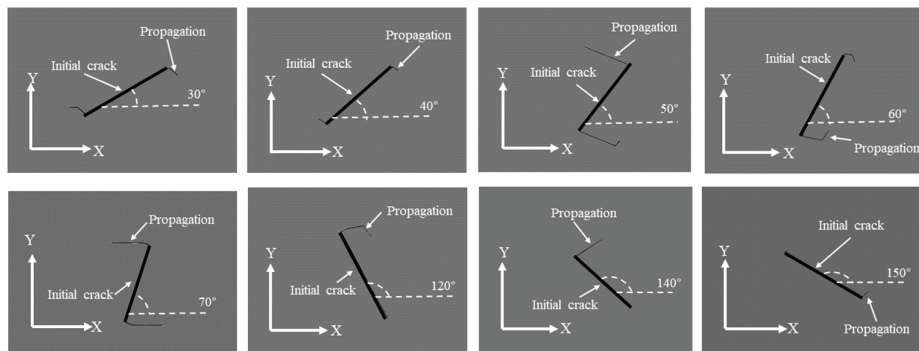


Figure 11. The crack propagation results under different pre-set inclinations for operational condition B.

The deformation coefficients in the resulting figures were set to 0 for ease of observing the crack extension. According to the crack propagation results of the pre-inserted cracks on the lower surface of the deflector trough under the two operating conditions, it was evident that with the continuous increase in the pre-set inclination angle, the crack propagation length exhibited a trend of initially increasing and then decreasing. The maximum crack propagation length was attained when the inclination angle approached 70° . Comparing the crack propagation lengths in the two operating conditions, it was observed that under the

same inclination angle and number of stress cycles, the crack propagation length in the region of maximum deformation was generally greater than in the region of maximum stress.

Figure 12 illustrates the relationship between the crack propagation direction and the location of the maximum principal stress at the crack tip. The black line represents the initially pre-set crack, while the thin blue line represents the newly formed portion due to fatigue crack extension. The green arrow indicates the direction of crack propagation, and the numbers above the arrows represent the element numbers penetrated by the crack. For example, “①” represents the number of the first unit cell penetrated by the crack. The red dots represent the positions of the maximum principal stress due to stress concentration when the crack penetrates each element, and the yellow dots represent the positions of the secondary stress (stress values just below the maximum principal stress) due to stress concentration when the crack penetrates each element. The numbers before the stress values correspond to the element numbers. For example, “1) 418 MPa” corresponds to a red value, and the digit “1” before the numerical value indicates the maximum stress value at the crack tip and its location when the crack penetrates the first unit cell. The adjacent pairs of points arranged side by side in the figure indicate that both the maximum principal stress and secondary stress occurred at that point before and after the crack penetration.

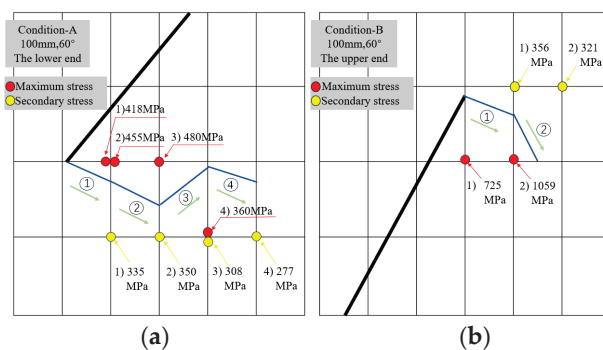


Figure 12. Diagram of crack propagation direction and principal stress locations. (a) Diagram of crack propagation in the lower region, (b) Diagram of crack propagation in the upper region.

Figure 12 shows that as the crack propagated within an element, stress concentrations occurred at the nodes along its advancement direction. The node's position with the maximum stress determined the deviation direction of the crack tip within that element. Simultaneously, the position and stress magnitude of secondary stress nodes influenced the deflection angle of the crack propagation.

The combined analysis of crack propagation trends under both operating conditions revealed that, under certain pre-set inclination angles (50° , 60°), crack propagation was not solely oriented towards increasing the length of the crack. After a certain distance of cracking, the crack trajectories at the upper and lower ends, or one of them, tended to extend into the region where the fracture had already occurred. This indicates a tendency to detach the surrounding structure from the lower surface of the deflector. If this trend of crack propagation continued, it would compromise the integrity of the deflector trough structure, reducing the strength of the lower surface structure. Simultaneously, detached fragments of the overall structure would be ejected with high kinetic energy, accompanying the plume flow field within the deflector trough. This could result in secondary damage to the deflector trough and other structures, posing a potential threat to the rocket engine. Therefore, particular attention should be given to the 50° and 60° inclination cracks originating from fatigue damage on the lower surface of the rocket deflector trough.

5.4. Minimum Initial Crack Length for Crack Propagation

Table 5 shows that the number of load cycles required for the initiation and propagation of pre-inserted cracks at different inclinations exhibited an overall trend of initially decreasing and then increasing with the increase in the inclination angle. For condition

A, the minimum initiation and propagation angle was 50° , while for condition B, the minimum initiation and propagation angle was 60° . In this section, the minimum initiation and propagation angles obtained from Table 5 for the two operating conditions were taken as pre-set angles to investigate the minimum initiation and propagation lengths of cracks at these two angles under their respective conditions.

Table 5. Crack initiation cycle numbers at different pre-set inclinations.

Inclination	30°	40°	50°	60°	70°	120°	140°	150°
Condition A	2317	6221	609	2191	1465	3013	5761	19,992
Condition B	32,217	31,647	1251	266	2719	3125	1435	23,495

This section explores the minimum initial length required for the continued expansion of the initial crack on the lower surface of the deflector groove. In this section, the variable being changed was the length of the initially pre-set crack on the rocket deflector groove. Regarding the practical scenario, within the service life cycle of a rocket launch platform, no more than 60,000 rocket launch missions and experiments will be undertaken. Therefore, with 60,000 load cycles as the upper limit, if no fatigue propagation occurs in the initial crack after 60,000 load cycles, it is considered that a crack of that length will not continue to propagate. This section also uses this criterion to determine whether the crack is propagating.

With the increase in the initial length of pre-existing cracks, the rate of fracture energy accumulation at the crack tip accelerates, reducing the number of stress cycle repetitions required for crack initiation. Figure 13 illustrates that, under condition A, the minimum crack initiation length was approximately 40 mm, whereas under condition B, it was around 20 mm. In the region of maximum stress, the number of cycles required for crack initiation is highly sensitive to changes in the initial crack length. As the initial crack length increases, the initiation cycles for cracks located at the point of maximum stress sharply decrease. In the region of maximum deformation, the increase in the initial crack length has a relatively weak impact on reducing the crack initiation cycles. The overall decrease remains relatively stable. From the figure, it can be observed that when the initial crack length exceeded 85 mm, cracks pre-existing in the region of maximum stress exhibited a faster crack initiation rate, whereas, for lengths below 85 mm, cracks situated in the region of maximum deformation demonstrated a faster initiation rate.

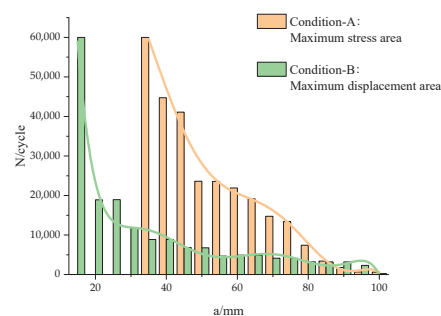


Figure 13. Load cycle number diagram before initiation of cracks with different lengths.

5.5. Crack Propagation Rate

Figure 14 shows that in both conditions, the crack propagation rate exhibited significant fluctuations and abrupt changes during the early stages of fatigue crack propagation. These fluctuations are attributed to variations in the crack propagation mechanism and changes in stress amplitude. Additionally, since ABAQUS cannot simulate stress concentration within the interior of elements, every crack propagation was bound to penetrate one or more element bodies completely, leading to fluctuations in the calculated crack propagation

rates. However, with an increase in the number of stress cycles, both the crack propagation rate and fluctuations eventually stabilised.

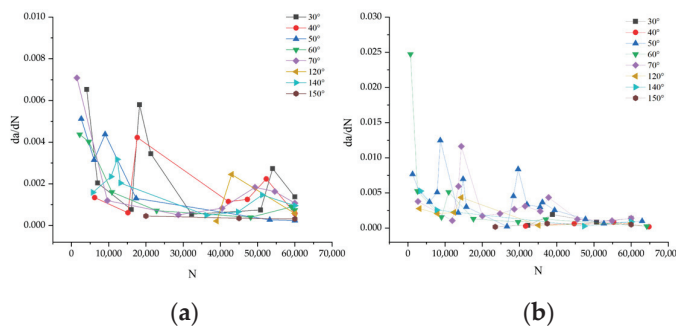


Figure 14. Crack propagation rate chart; (a) Condition A; (b) Condition B.

By comparing the two operating conditions, it can be observed that the fluctuation amplitude of the crack propagation rate under condition A was approximately 0.0063, which is smaller than the fluctuation amplitude under condition B (0.022). Figure 14 shows that the crack propagation rate in the region of maximum deformation was higher than that in the region of maximum stress. Simultaneously, the crack in the region of maximum deformation reached the stable propagation stage faster compared to the region of maximum stress. This is because the region of maximum stress undergoes larger stress gradient changes during the loading process, leading to significant differences in stress gradients at the ends of the crack. This results in a smaller stress ratio, exacerbating the amplitude variations in cyclic stress, and thereby prolonging the time for the crack propagation rate to stabilise.

5.6. Analysis of the Influence of High-Temperature Environment on Fatigue Life

To investigate the impact of temperature on the fatigue crack propagation of the lower surface of the deflector trough, it was necessary first to study the influence of temperature loads on the stress distribution of the lower surface.

Figure 15 presents the stress contour map of the lower surface of the rocket deflector at a specific moment under the influence of temperature loads or without them. As shown in the diagram, under the action of a single impact force, the maximum stress on the lower surface of the deflector was located in the middle region directly impacted by the tail flame, with a peak value of 175 MPa. However, when coupled with temperature loads, the concentration of stress on the lower surface increased, and it was distributed more in the upper-middle region. The maximum stress value rose to 198 MPa, representing an increase of approximately 13%.

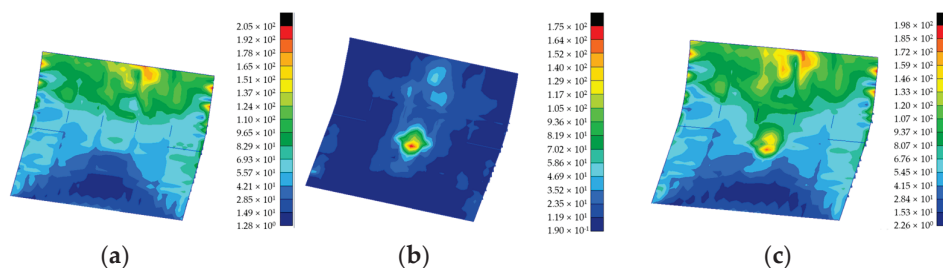


Figure 15. (a) Temperature load, (b) Impact force load, (c) Coupled load.

According to Table 6, it can be observed that in the early stages of rocket launch, as the launch time progressed, the impact of coupled temperature load on the increase in stress on the lower surface of the deflector trough exhibited a gradually decreasing trend. When the rocket tail flame jet's distribution state stabilised on the deflector trough's lower surface, the temperature load increased the maximum stress value on the lower surface of the

deflector trough, which was subjected only to a single impact force, by approximately 10%. Through Figure 15, it can be observed that temperature load altered the stress distribution on the lower surface of the deflector under the action of a single exhaust plume impact load, causing the concentration of stress to shift from the central region directly impacted by the rocket's tail flame to the upper-middle region. The maximum stress value induced by the temperature load (205 MPa) was greater than the maximum stress values under the exhaust plume impact load alone (175 MPa) and the coupled load (198 MPa). This indicates that the coupling of temperature and exhaust plume impact loads decreases the maximum stress on the lower surface, thereby enhancing the fatigue life of the deflector's lower surface.

Table 6. Influence of temperature load.

T/s	0.14	0.28	0.42	0.70	0.85	0.98	1.1
Impact Force Load (MPa)	10.3	32.4	70.0	129.0	157	183.0	175.0
Coupled Load (MPa)	221.0	201.0	174.0	182.0	185.0	190.0	198.0
Impact of Temperature Load (%)	95.3	83.9	60.0	29.1	15.1	3.7	11.6

The impact of temperature load = (coupled load – exhaust plume impact load)/coupled load.

Applying the S-N curve method for calculating the crack initiation life on the lower surface of the deflector revealed that cyclic loading from a single impact force alone did not lead to structural fatigue failure on the lower surface. Consequently, there was no initiation or propagation of cracks on the lower surface of the deflector under the influence of an exhaust plume impact load alone.

6. Conclusions

This paper, based on the XFEM, draws the following conclusions through a study on fatigue crack propagation on the lower surface of an offshore rocket deflector:

- (1) Under the coupling effect of high temperature and impact, the length of fatigue crack propagation on the lower surface of the rocket deflector exhibits an increasing trend with the increase in the pre-set tilt angle, followed by a decrease when the tilt angle exceeds 70°. The maximum crack propagation length is reached when the tilt angle approaches 70°.
- (2) The node's position with the maximum circumferential stress at the crack tip will determine the crack propagation direction. Simultaneously, the position and magnitude of secondary stress nodes will influence the deflection angle of crack propagation.
- (3) Compared to the exhaust plume impact load alone, the coupled temperature load not only increases the maximum stress on the lower surface of the deflector by approximately 10% but also shifts the stress concentration region from the central area directly impacted by the rocket's tail flame to the upper-middle region. This, in turn, promotes the initiation and propagation of cracks on the lower surface of the deflector.

Author Contributions: Conceptualization, Z.X. and Y.Y.; methodology, Z.X.; software, C.Z.; validation, T.L.; formal analysis, C.Z.; investigation, R.L.; resources, Z.X.; data curation, T.L. and Y.Y.; writing—original draft preparation, Z.X. and C.Z.; writing—review and editing, Z.X., C.Z. and Y.Y.; visualization, R.L. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Acknowledgments: We are grateful for the software support from the Marine Design and Research Institute of China.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Cui, W.; Cai, X.; Leng, J. A state-of-the-art review for the fatigue strength. *J. Ship Mech.* **1998**, *4*, 63–81.
2. Bergara, A.; Dorado, J.I.; Martín-Meizoso, A.; Martínez-Esnaola, J.M. Fatigue crack propagation in complex stress fields: Experiments and numerical simulations using the Extended Finite Element Method (XFEM). *Int. J. Fatigue* **2017**, *103*, 112–121. [CrossRef]
3. Belytschko, T.; Black, T. Elastic crack growth in finite elements with minimal remeshing. *Int. J. Numer. Methods Eng.* **1999**, *45*, 601–620. [CrossRef]
4. Sukumar, N.; Möes, N.; Moran, B.; Belytschko, T. Extended finite element method for three-dimensional crack modeling. *Int. J. Numer. Methods Eng.* **2000**, *48*, 1549–1570. [CrossRef]
5. Sukumar, N.; Chopp, D.L.; Moran, B. Extended finite element method and fast marching method for three-dimensional fatigue crack propagation. *Eng. Fract. Mech.* **2003**, *70*, 29–48. [CrossRef]
6. Patzák, B.; Jirásek, M. Process zone resolution by extended finite elements. *Eng. Fract. Mech.* **2003**, *70*, 957–977. [CrossRef]
7. Zhang, J.; He, X.D.; Sha, Y.; Du, S.Y. The compressive stress effect on fatigue crack growth under tension–compression loading. *Int. J. Fatigue* **2010**, *32*, 361–367. [CrossRef]
8. Menouillard, T.; Réthoré, J.; Combescure, A.; Bung, H. Efficient explicit time stepping for the eXtended Finite Element Method (X-FEM). *Int. J. Numer. Methods Eng.* **2006**, *68*, 911–939. [CrossRef]
9. Fries, T.-P.; Baydoun, M. Crack propagation with the extended finite element method and a hybrid explicit-implicit crack description. *Int. J. Numer. Methods Eng.* **2011**, *89*, 1527–1558. [CrossRef]
10. Daux, C.; Möes, N.; Dolbow, J.; Sukumar, N.; Belytschko, T. Arbitrary branched and intersecting cracks with the eXtended Finite Element Method. *Int. J. Numer. Methods Eng.* **2000**, *48*, 1741–1760. [CrossRef]
11. Pathak, H.; Singh, A.; Singh, I.V.; Yadav, S.K. Fatigue crack growth simulations of 3-D linear elastic cracks under thermal load by XFEM. *Front. Struct. Civ. Eng.* **2015**, *9*, 359–382. [CrossRef]
12. Loehnert, S.; Mueller-Hoeppe, D.S.; Wriggers, P. 3D corrected XFEM approach and extension to finite deformation theory. *Int. J. Numer. Methods Eng.* **2011**, *86*, 431–452. [CrossRef]
13. Sutula, D.; Kerfriden, P.; Van Dam, T.; Bordas, S.P.A. Minimum energy multiple crack propagation. Part-II: Discrete solution with XFEM. *Eng. Fract. Mech.* **2018**, *191*, 225–256. [CrossRef]
14. Fleck, N.A.; Shin, C.S.; Smith, R.A. Fatigue crack growth under compressive loading. *Eng. Fract. Mech.* **1985**, *21*, 173–185. [CrossRef]
15. Sahouryeh, E.; Dyskin, A.V.; Germanovich, L.N. Crack growth under biaxial compression. *Eng. Fract. Mech.* **2002**, *69*, 2187–2198. [CrossRef]
16. Nikfam, M.R.; Zeinoddini, M.; Aghebat, F.; Arghaei, A.A. Experimental and XFEM modelling of high cycle fatigue crack growth in steel welded T-joints. *Int. J. Mech. Sci.* **2019**, *154*, 178–193. [CrossRef]
17. Teng, Z.H.; Liao, D.M.; Wu, S.C.; Sun, F.; Chen, T.; Zhang, Z.B. An adaptively refined XFEM for the dynamic fracture problems with micro-defects. *Theor. Appl. Fract. Mech.* **2019**, *103*, 102255. [CrossRef]
18. Mazlan, S.; Yidris, N.; Koor, S.S.R.; Petru, M. Experimental and numerical analysis of fatigue life of aluminum Al 2024-T351 at elevated temperature. *Metals* **2020**, *10*, 1581. [CrossRef]
19. Abdollahnia, H.; Alizadeh Elizei, M.H.; Reza Kashyzadeh, K. Multiaxial fatigue life assessment of integral concrete bridge with a real-scale and complicated geometry due to the simultaneous effects of temperature variations and sea waves clash. *J. Mar. Sci. Eng.* **2021**, *9*, 1433. [CrossRef]
20. Ou, M.; Cao, W. On the mechanic performance of building-purpose Q235 steel under high-temperature (Fire). *J. Zhuzhou Inst. Technol.* **2006**, 99–101.
21. Xiao, M. Testing for paris coefficients C_m and ΔK_{th} of Q235. *J. Mech. Strength.* **2003**, *2*, 215–218. [CrossRef]
22. Jin, S.; Zhang, W. Analysis and verification for the crack growth of lower-wing panel based on XFEM. *J. Xi'an Aeronaut. Inst.* **2023**, *41*, 18–23. [CrossRef]
23. He, L.; Liu, Z.; Gu, J.; Wang, J.; Men, K. Fatigue crack propagation path and life prediction based on XFEM. *J. Northwest. Polytech. Univ.* **2019**, *37*, 737–743. [CrossRef]
24. Wang, Z. Crack Propagation of Coiled Tubing Based on Extended Finite Element Method. Master's Thesis, Yangtze University, Jingzhou, China, 2019.
25. Li, Q. Aeroengine Turbine Disk GH4133B Alloy Fatigue Crack Propagation Numerical Simulation Study. Master's Thesis, Xiangtan University, Xiangtan, China, 2018.
26. Melenk, J.; Babuska, I. The partition of unity finite element method: Basic theory and applications. *Comput. Methods Appl. Mech. Eng.* **1996**, *139*, 289–314. [CrossRef]
27. Sukumar, N.; Chopp, D.L.; Möes, N.; Belytschko, T. Modelling holes and inclusions by level sets in the extended finite element method. *Comput. Methods Appl. Mech. Eng.* **2001**, *190*, 6183–6200. [CrossRef]

28. Ventura, G.; Budyn, E.; Belytschko, T. Vector level sets for description of propagating cracks in finite elements. *Int. J. Numer. Methods Eng.* **2003**, *58*, 1571–1592. [CrossRef]
29. Gledić, I.; Mikulić, A.; Parunov, J. Improvement of the ship emergency response procedure in case of collision accident considering crack propagation during salvage period. *J. Mar. Sci. Eng.* **2021**, *9*, 737. [CrossRef]
30. Killpack, M.; Abed-Meraim, F. Limit-point buckling analyses using solid, shell and solid–shell elements. *J. Mech. Sci. Tech.* **2011**, *25*, 1105–1117. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Experimental and Numerical Analysis of Supporting Forces and Lashing Forces in a Ship Cargo Securing Scheme

Mengxiang Li ^{1,†}, Guo Wang ^{2,†}, Kun Liu ^{1,*}, Yue Lu ¹ and Jiaxia Wang ¹

¹ School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang 212003, China; limengxiang_just@163.com (M.L.); yuelu@just.edu.cn (Y.L.); jxwang66@yeah.net (J.W.)

² Marine Design and Research Institute of China, Shanghai 200011, China; wg198885@126.com

* Correspondence: kunliu@just.edu.cn; Tel.: +86-135-1169-2085; Fax: +86-0511-8444-6543

† These authors contributed equally to this work.

Abstract: The safety assessment of ship cargo securing systems is of significant importance in preventing casualties, vessel instability, and economic losses resulting from the failure of securing systems during transportation in adverse sea conditions. In this study, an independently designed cylindrical cargo securing scheme with supporting structures was adopted for investigation. Utilizing a sway device, three-degree-of-freedom coupled motion encountered during ship transportation was obtained, and data regarding changes in the support forces at the foundations and tension forces in the lashing ropes were collected. Subsequently, numerical simulations were conducted using the multibody dynamics software ADAMS 2020. The results obtained from the simulations were compared with the experimental data. The overall tendencies were accurately predicted in the numerical analysis. It was observed that the difference of the peak support forces between the numerical simulation results and the experimental data were within a 10% margin. In terms of the lashing ropes, the difference was limited, within 9%. These findings demonstrate that numerical simulation techniques can provide valuable insights for verifying the safety of practical cargo securing systems.

Keywords: shipping; cargo securing; ADAMS; lashing force; support force

1. Introduction

During transportation, the securement of cargo and equipment has consistently been recognized as a significant influencing factor affecting vessel navigation and cargo safety. If the cargo securing method is not appropriate, the cargo may move to one side, increasing the risk of capsizing when the ship's course changes rapidly [1]. When encountering adverse sea conditions, the violent motion of the vessel can lead to the failure of container securing systems, resulting in significant losses, such as container collapse, ship instability and capsizing, and casualties [2]. Vehicles and cargo, whether on the ship's deck or in the cargo hold, require the implementation of securement measures to prevent cargo displacement and tipping during transport due to unexpected adverse weather conditions such as sudden waves and strong winds. However, there are limited stipulations within the relevant securing documentation concerning stowage techniques for non-standard cargo. In practical operational procedures, cargo securing is often executed based on the crew's own experiences, lacking clear corresponding securing standard verification requirements. This results in inefficient securing practices, leading to occurrences of either excessive or insufficient restraints [3,4]. With the development of maritime transportation, the demand for enhanced safety in cargo securing methods has become increasingly urgent, prompting scholarly focus on the safety assessment of securing cargo on ship decks.

Some scholars employ experimental research to measure the forces acting on ship-borne cargo during actual transportation processes. Mikael Sternsso and Bjorkenstam [5]

utilized vehicles equipped for long-term measurement of lashing forces and deck's reaction forces on the Ro-Ro vessel's deck. Their findings revealed limitations in traditional vehicle securing methods, with lateral displacement primarily governed by friction forces between tires and the deck. Junbo Jia [6] conducted experimental research on the frictional forces experienced by automobile tires on Ro-Ro vessels, emphasizing the safety concerns related to lateral displacement and sliding. In subsequent research, Junbo Jia [7] proposed that the safety of vehicle transportation is primarily influenced by the oscillatory motion of ships, and the motion of the ship's hull is highly correlated with wave height and loading conditions. Under specific weather conditions and on certain routes with favorable sea conditions, vehicles can be securely positioned in most areas of the ship without being fastened. Matusiak [8] also conducted simulations of unfastened cargo in irregular wave conditions at an earlier stage, recognizing the indispensability of securing schemes due to motion-induced sliding forces exceeding available friction. The Swedish Transport Administration, in collaboration with the Swedish company MariTerm AB, conducted extensive tensile rupture tests on chains and lashing straps. It was observed that the strength of lashing straps significantly decreases with prolonged usage, whereas the strength of chains remains relatively unchanged. However, when utilizing chains for securing, there is a higher probability of rupture occurring at the chain tensioner location. Therefore, special attention should be given to the strength of chains at the chain tensioner location [9].

Experimental research provides relatively accurate and realistic feedback on cargo securing conditions under actual ship navigation motions. However, experimental research typically involves lengthy cycles and requires a significant investment of manpower and resources, so the verification of securing schemes for shipping companies can be cost-intensive. The rapid development of finite element simulation technology and multibody dynamics simulation technology has facilitated the study of cargo securing on ships. Turnbull [10] simplified the suspension, tires, and supports of semi-trailers and established a numerical model for securing semi-trailers on the deck of Ro-Ro vessels. They conducted a detailed analysis of the forces experienced by the lashing ropes of vehicles during the overall roll and pitch motions of the Ro-Ro vessel. Surendran [11] established a numerical model of ship motion to ensure safety during navigation. The research extensively investigated the swaying issues of Ro-Ro vessels in waves and explored strategies to enhance ship stability when encountering resonance conditions with waves. Kawamura [12] utilized GPU to conduct Smoothed Particle Hydrodynamics (SPH) simulations of the six degrees of freedom (6DoF) ship motion under maritime transport conditions. Subsequently, a comparative analysis was performed by comparing the SPH simulation results with actual test data to investigate the predictive accuracy of the SPH method. Jiaqi Liu, Chuntong Li, Deyu Wang [13–16] utilized numerical simulation software to simulate container lashing bridges. They conducted comparative analysis by performing scaled-down model tests and found a high degree of concordance between the calculated results and the experimental outcomes. This indicates that existing numerical simulation techniques can effectively enhance design efficiency and precision, possessing a degree of generality. Fangzhen Song [17] utilized ADAMS software to construct a dynamic model and employed spring forces to connect the ship's hull model, simulating buoyancy on the water surface. This was done to verify the role of ballast water in maintaining the stability of the ship's hull. Edwin Kreuzer [18] has developed a software that utilizes the principles of multibody dynamics to compute the lashing forces of cargo securing systems on Ro-Ro ships. This software permits the verification of the safety of the lashing system by inputting the vessel's motion, offering a convenient tool for shipping companies. This software serves as a convenient tool for shipyards and suppliers to perform efficient checks on lashing securing systems. Nam Woongshik [19] employed ABAQUS finite element analysis to predict the feasibility of using D-ring devices in lashing securing systems. Zhu Hong-jun [20] established a dynamic model of a ship-loading and unloading device using ADAMS software, proposing structural optimization design schemes based on forces acting on wheels and lashing straps.

For the securement of non-standard goods, *Code of Safe Practice for Cargo Stowage and Securing* (CSS Code) provides provisions on securing schemes, equipment, and strength. On this basis, Jerzy Kabacinski [3] proved that simplification in the standard calculation affects the safety margin of non-standard goods and proposed a more reliable calculation method without simplification. Ling De-chao [21] also started from the specification and improved the horizontal and vertical balance checking methods of non-standardized cargo units, considering meteorological conditions. The specification calculation can provide a reference for the safety verification of cargo securing. However, the focus of the specification is on the strength of the securing rigging, and there are few studies on the influence of the load on the cargo itself. At the same time, the calculation method of the specification simplifies the actual situation, affecting the reliability of the calculation results.

Utilizing numerical simulation techniques can effectively enhance safety assessments for securing solutions, offering valuable insights for their design and optimization. However, the reliability of simulation calculations requires further research. Previous studies have primarily focused on investigating whether the lashing forces on the securing system during transportation exceed the allowable loads for lashing. However, with the differentiation of transported goods, the impact of the lashing system on the load of the secured cargo should also be taken into consideration.

This study presents a proprietary securing scheme that utilizes pedestal supports and lashing ropes to secure a cylindrical cargo, with the aim of investigating the characteristics of lashing rope forces and pedestal support forces in the securing scheme. A three-degree-of-freedom coupled motion swing experiment was conducted using a dedicated three-degree-of-freedom sway test rig. Subsequently, numerical simulations of the experimental conditions were performed using the multibody dynamics software ADAMS. A comparison between the tension data of the lashing ropes and the pedestal support forces obtained from both methods indicated a favorable agreement. These findings suggest that the simulation results can serve as a reliable reference for future numerical assessments of the safety of cargo securing schemes.

2. Experimental Investigations

2.1. Test Model and Arrangement

In this study, the sway test was conducted using a three-degree-of-freedom swing test apparatus, as depicted in Figure 1. The overall structure of the device consists of three main components: the sway table base, the motion device, and the securing and lashing platform. The carrying platform of this test rig measures 2 m × 4 m, with a maximum load capacity of 3 t. It is capable of simulating multi-degree-of-freedom coupled motion with angular ranges of $\pm 30^\circ$ in roll, $\pm 15^\circ$ in pitch, and vertical displacement of ± 0.35 m, thereby replicating the roll, pitch, and heave motions of the securing system during a ship's motion.

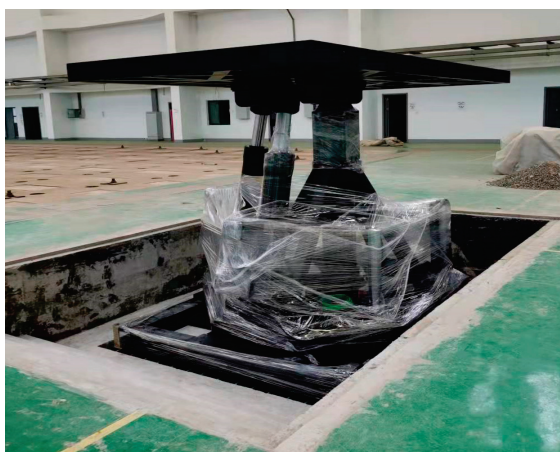


Figure 1. Three-degree-of-freedom swing test apparatus.

In this study, a similar securing scheme was designed with reference to the placement of the wind turbine tower. It is assumed that the fixed object is a cylindrical structure supported by four pedestals. In order to prevent the cylinder from falling off the pedestals, the lateral spacing of the pedestals is increased. The bracing part of the pedestal points toward the center of the cross section of the cylinder, forming a 57° angle with the vertical direction. Use the rope to reload the upper side of the cylinder to complete the overall securing. See Figure 2.

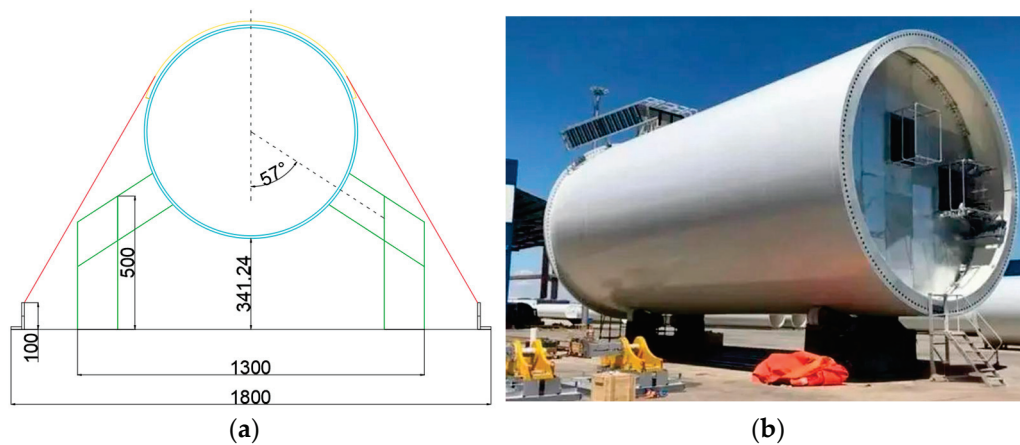
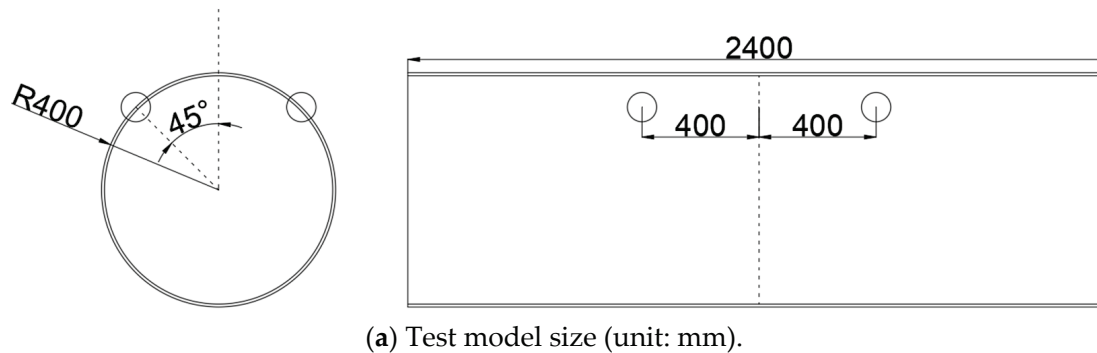


Figure 2. (a) Securing scheme side view. (b) Tower placement schematic diagram.

The primary model used in the experiment is a thin-walled cylindrical structure with a length of 2400 mm, an outer radius of 400 mm, and a wall thickness of 10 mm. The overall mass of the model is 465 kg, and lifting lugs are welded on the top for hoisting, relocation, and safety purposes. See Figure 3.



(a) Test model size (unit: mm).



(b) Test model photograph

Figure 3. Test model.

The cylindrical model is placed on four symmetrically arranged L-shaped support pedestals, constructed using 20 mm-thick steel plates. The supporting surfaces of the pedestals are inclined at an angle of 57° with respect to the vertical axis. The steel plates on the supporting surfaces are curved to facilitate a better fit with the test model. The bottom of the support pedestals is welded to the surface of the sway table at predetermined positions. The dimensions and actual installation of the support pedestals are depicted in Figure 4.

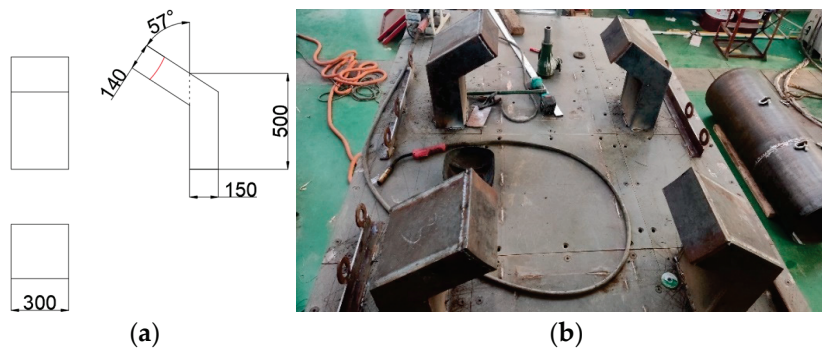


Figure 4. (a) Support pedestal model dimensions. (b) Support pedestal site model installation.

After placement, rubber pads are positioned above the cylindrical model, and lashing ropes are arranged in a cross pattern on top of the rubber pads for preload fixation, as shown in Figure 5.



Figure 5. Tie rope ballast arrangement.

In this experiment, the center of the sway table securing platform plane is considered as the origin point, with the x -axis defined along the longitudinal direction of the sway table, the y -axis along the transverse direction, and the z -axis representing the vertical direction.

As shown in Figure 6, the blue wireframe represents the test model, the green wireframe represents the support pedestals, the yellow wireframe represents the rubber pads, and the red lines represent the lashing ropes used for cross preload fixation. In this experiment, two lashing configurations were selected: the middle condition and the offset condition. In the middle condition, the cylindrical model is positioned at the center of the platform. The four pedestals are divided into two groups and symmetrically arranged about the y -axis in the XY plane. The distance between the two groups of pedestals in the

x -axis direction is 1100 mm. Above the cylindrical model, three rubber pads of dimensions 1000 mm $\times$ 300 mm $\times$ 10 mm are symmetrically placed, with a spacing of 450 mm between each pad. Two nylon ropes are symmetrically crossed and fixed on each rubber pad, with their intersection at the center of the rubber pad. The two ends of the ropes are secured to lashing components on the platform, which are symmetrically arranged about the x -axis, with a spacing of 1800 mm. The distance between the lashing points on the same side of the rubber pad is 500 mm, and the lashing points are positioned 140 mm above the platform surface. The pedestal placement location shifts towards the negative x -axis direction by 450 mm compared to the centered placement, while all other component placement positions remain the same.

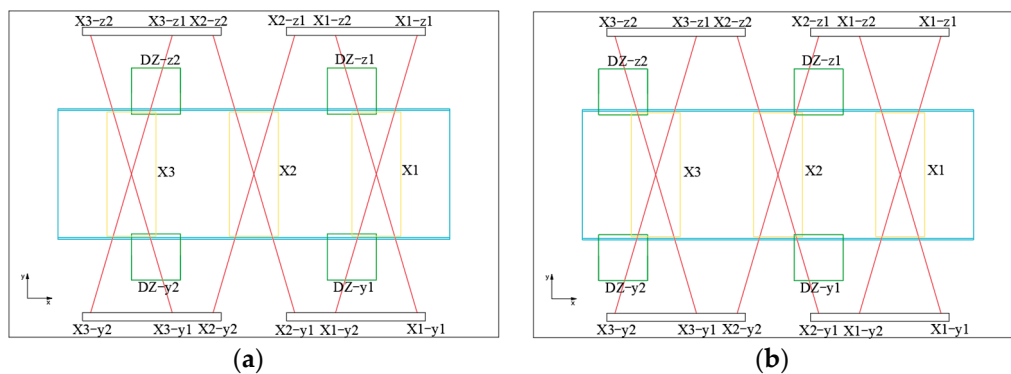


Figure 6. (a) Middle condition. (b) Offset condition.

The x -axis positive direction is defined from left to right, while the y -axis positive direction is on the left side and the y -axis negative direction is on the right side. The positions of the rubber pads along the negative x -axis direction are ordered and coded as X1, X2, X3. The lashing rope securing points are encoded with reference to the rubber pad coding and follow the format ‘Rubber Pad Code—Securing Point Position Information’. The securing points on the left side of the respective rubber pads on the negative x -axis direction are sequentially labeled as z1, z2, while those on the right side are labeled as y1, y2. For example, ‘X2-y2’ represents the second securing point on the right side of the second rubber pad when viewed from the negative x -axis direction. The support pedestal coding uses ‘DZ’ to denote the pedestal, ‘y’ for the right side of the test model, and ‘z’ for the left side of the test model. They are sequentially numbered as 1 and 2 on the negative x -axis direction. For instance, ‘DZ-z2’ indicates the second pedestal on the left side of the test model when viewed from the negative x -axis direction.

2.2. Selection of Test Conditions

In order to assess the influence of different motion parameters on various lashing methods and to conduct an in-depth analysis of the load variations in lashing components, the following experimental scenarios have been established in conjunction with the test conditions, as summarized in Table 1.

Table 1. Summary of test conditions.

IDX	Condition Number	Description of Working Condition
1	M1	Middle condition, group 1 parameter
2	M2	Middle condition, group 2 parameter
3	M3	Middle condition, group 3 parameter
4	O1	Offset condition, group 1 parameter
5	O2	Offset condition, group 2 parameter
6	O3	Offset condition, group 3 parameter

According to the test conditions, M/O represents the middle and offset conditions, respectively, and 1/2/3 represents different preset motion parameter groups. The motion parameters are shown in Table 2.

Table 2. Parameter settings of motion parameter groups.

Group ID	Roll Amplitude	Roll Period	Pitch Amplitude	Pitch Period	Heave Amplitude	Heave Period
1	10°	10 s	5°	12 s	100 mm	8 s
2	20°	10 s	10°	12 s	200 mm	8 s
3	25°	10 s	15°	12 s	300 mm	8 s

2.3. Test Data Acquisition

Figure 7 illustrates the arrangement of membrane pressure sensors and tension sensors used in the experiment. The cylindrical model simulates cargo that is supported.

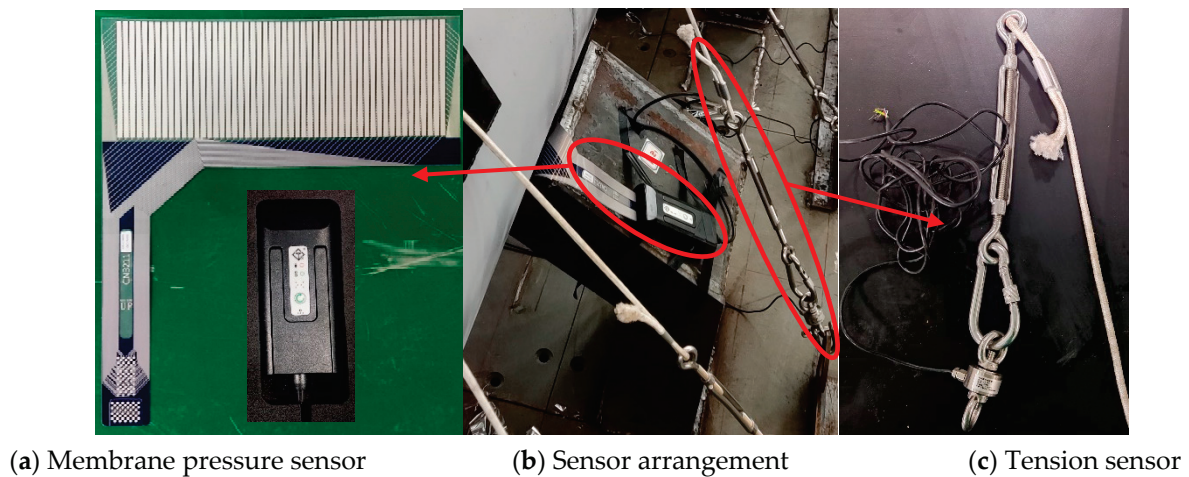


Figure 7. The arrangement of membrane pressure sensor and tension sensor.

Each support pedestal is equipped with rubber blocks and membrane pressure sensors on the support surface. The rubber blocks are slightly smaller than the sensing area of the membrane pressure sensors to ensure that the contact area is within the sensing region. The sensors measure all the pressures on the contact surface, and the numerical values of the support forces are directly recorded through dynamic signal acquisition, documenting the load changes of each support pedestal under different motion postures during the experiment. The lashing forces during the experiment are collected using tension sensors. Tension sensors are positioned on both sides of each lashing rope, near the lashing point locations. The sensors are connected to the basket screws through loops, and the lashing ropes are secured to the other end of the basket screw. Prior to the start of the experiment, the tension in each lashing rope is adjusted to 200 N by adjusting the basket screws. After the arrangement is completed, the dynamic signal acquisition allows for the direct recording of the changes in lashing forces. The two overall arrangements for the experiment are shown in Figure 8.



Figure 8. Overall layout of (a) middle condition; (b) offset condition.

2.4. Partial Test Data

Figures 9 and 10 show the time-history curves of support forces and lashing rope loads for motion conditions M3 and O3 during the experiment. In the centered condition, due to the symmetric arrangement of components, the support forces of each pedestal oscillate within a relatively concentrated range. However, in the offset condition, as the pedestals are biased toward the rear, there are significant differences in the support forces between the front and rear groups of pedestals. The front pedestals, being closer to the center of gravity of the cylinder, experience much higher support forces compared to the rear pedestals. Additionally, in the offset condition, the first group of lashing ropes is subjected to a larger relative displacement due to the cylinder's overhang position, resulting in greater variations in tension in this group compared to the other two. When comparing the two conditions, although there are significant differences in the distribution of support forces due to the varying positions, the resultant forces are relatively similar since the model's motion posture remains consistent under the same motion parameters. Due to the differences in the periods of motion across different degrees of freedom, variations in phase differences occur during motion, leading to periodic changes in peak loads for various components in the whole movement.

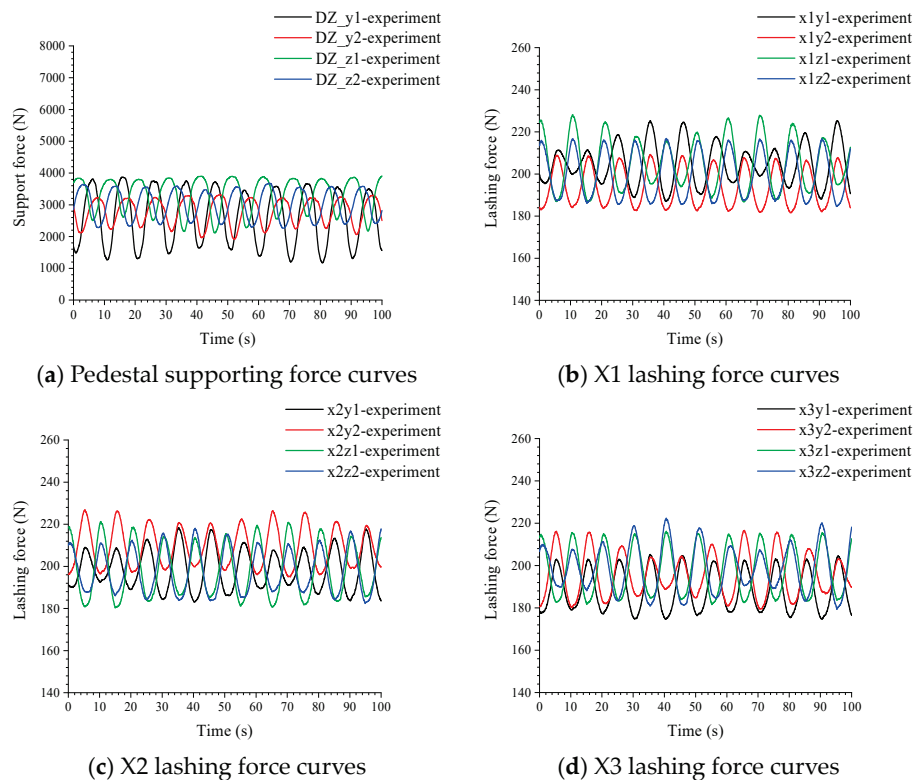


Figure 9. M3 load curve of each component under test condition.

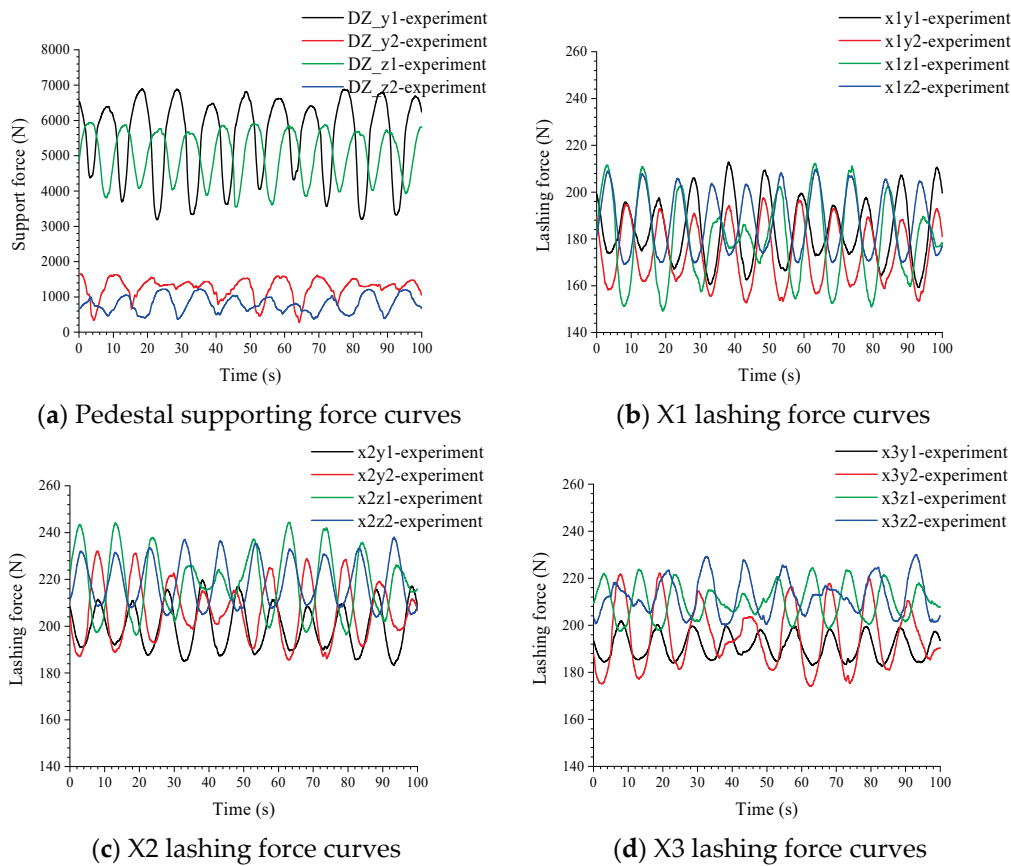


Figure 10. O3 load curve of each component under test conditions.

3. Numerical Simulation

In this section, based on the model test conditions described above, a simulation analysis of the swinging motion of the model under lashing states was conducted using the dynamics software ADAMS. This analysis resulted in time–history curves for the loads on each support pedestal and lashing ropes during the motion. These simulated results were compared with the experimental results to validate the numerical simulation technique.

3.1. Numerical Simulation Model

Figure 11a represents the overall computational models for both lashing configurations. When conducting simulation analyses of the model's lashing experiment, the focus was on the variation of support pedestal forces and lashing rope loads during the overall motion. The overall motion can be achieved by applying driving functions to the platform model. Therefore, the motion device below the platform is omitted in the modeling, and models are only established for the platform and all components above it. Figure 11b illustrates the simulation model for the test model, which is consistent with the experimental model and made of ordinary steel. Figure 11c shows the numerical simulation model for the swinging device platform, where a driving function was applied to simulate relative swinging motion with respect to the ground, replicating the actual test platform's motion. Figure 11d represents the numerical simulation model for the support pedestals, with each pedestal individually modeled and fixed relative to the swinging device platform to simulate the pedestals welded to the platform surface. Figure 11e shows the simulation model for the lashing ropes' compression loading method. Due to the limitation of ADAMS in modeling spring forces as point-to-point linear connections, it cannot directly simulate the bending compression state of the lashing ropes. Therefore, a cross-shaped block was created above the rubber pad to represent the lashing rope positions, and these blocks were connected at

both ends to the corresponding lashing points to simulate the lashing ropes' compression loading.

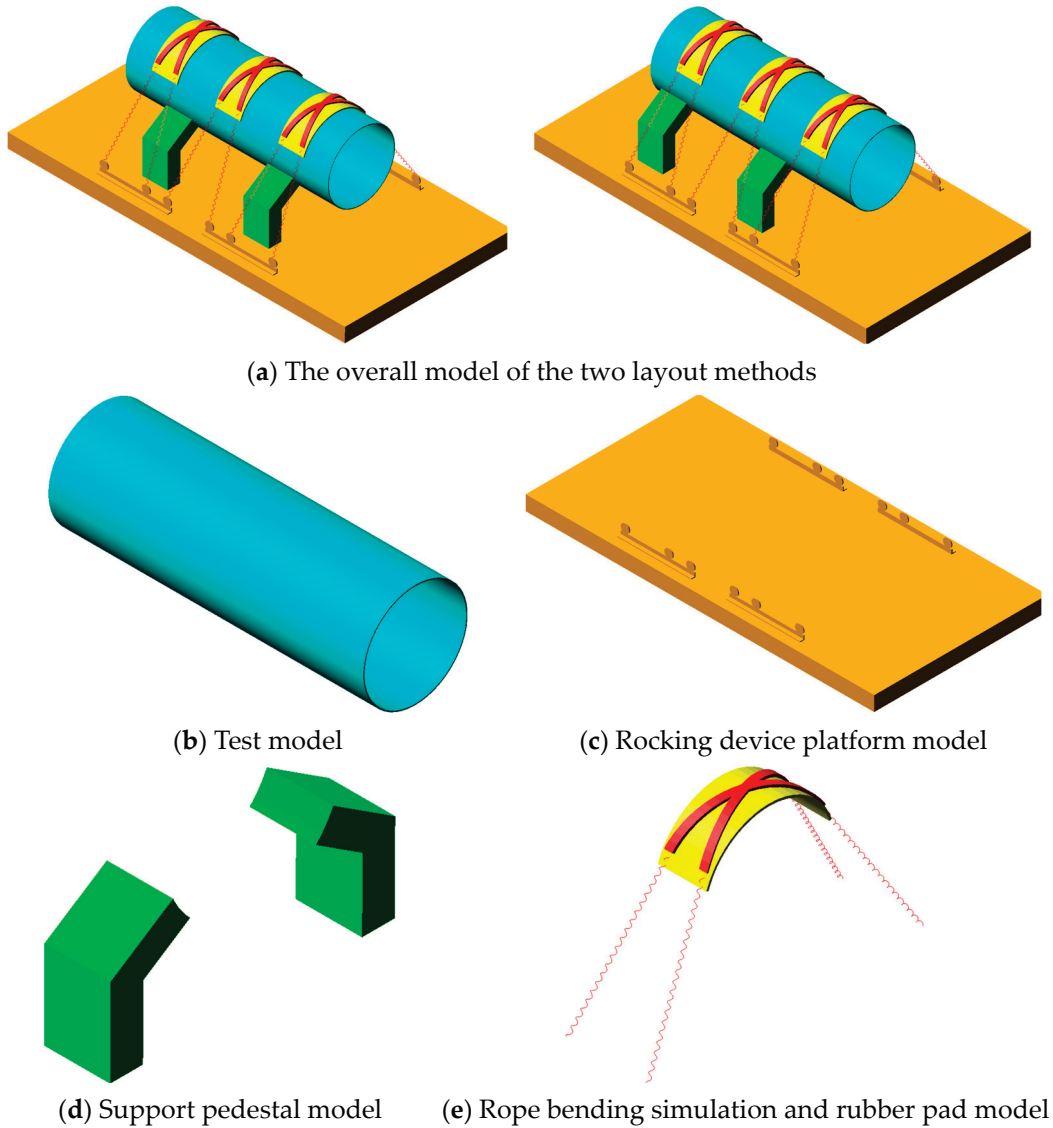


Figure 11. Each component model in simulation calculation.

3.2. Simulation Method

3.2.1. Contact Parameters

To individually output the support forces for each support pedestal in the modeling process, the models of the four support pedestals were connected to the platform using fixed joints. Contact settings in the Forces module were used to set the Coulomb friction coefficients and contact parameters for the overall simulation model. Since rubber pads were placed on the contact surfaces of the support pedestals, the contact between the support pedestals and the test model was configured based on the interaction between rubber and steel. The parameters for contact forces were set based on classical theoretical calculations [22].

According to Hertz contact theory, the relationship between contact normal force P and the deformation δ can be derived from:

$$\delta = \frac{a^2}{R} = \left(\frac{9P^2}{16RE^{*2}} \right)^{\frac{1}{3}}$$

It can be deduced that:

$$P = K\delta^{\frac{3}{2}}$$

In this Equation, K depends on the contact object material and structure shape.

$$K = \frac{4}{3}R^{\frac{1}{2}}E^*$$

where $\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2}$. R_1, R_2 are the contact radius of contact objects at the contact point. $\frac{1}{E^*} = \frac{(1-\mu_1^2)}{E_1} + \frac{(1-\mu_2^2)}{E_2}$. μ_1, μ_2 are Poisson's ratio of two contact objects' materials. E_1 and E_2 are modulus of elasticity of two contact objects' materials.

According to the *Mechanical Design Manual*, the static friction coefficient of rubber to steel is 0.8, and the dynamic friction coefficient is 0.5.

These settings allowed for the simulation of contact interactions between the support pedestals and the test model, enabling the calculation and output of support forces for each individual pedestal during the simulation.

3.2.2. Lashing Stiffness

As the ropes only experience tension and not compression, the relationship between spring force and spring length variation was defined using spline parameters in the Forces module of the software. This was done in a way that ensures the spring force is zero when the rope length is less than the initial value. During the spring setup process, an initial spring length shorter than the securing rope spacing was set to apply the pre-tension force to the ropes.

The stiffness value of the ropes was determined through tensile testing. Tensile testing of the nylon ropes was conducted on a universal testing machine, as shown in Figure 12. The obtained stress-strain curve from the tensile test is illustrated in Figure 13. It was observed that the force-displacement curve of the 500 mm-long nylon rope exhibited linear behavior. Through linear fitting, it was determined that the stiffness value of the 500 mm nylon rope is 85.09 N/mm. Subsequently, the corresponding stiffness values were calculated for the actual lengths of nylon ropes used and defined in ADAMS for the spring forces.

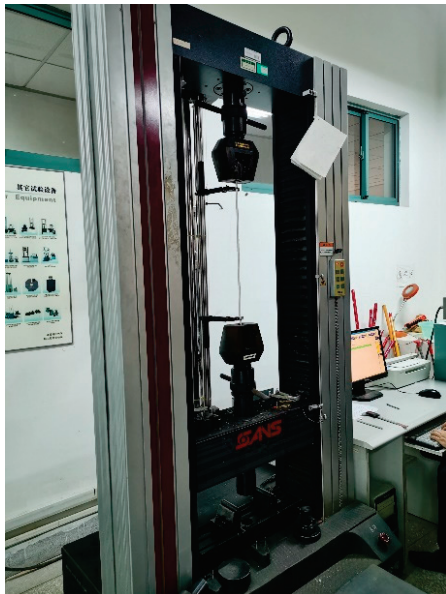


Figure 12. Nylon rope tensile test.

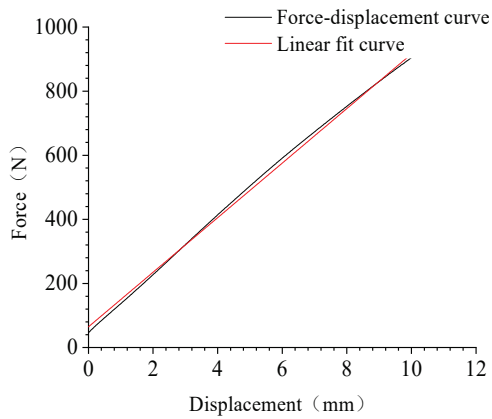


Figure 13. Tensile test results for nylon rope.

3.2.3. Swing Device Platform Motion

The rotation of the swinging platform relative to the reference point (pivot) was defined using the Revolute joint connector within ADAMS software's Connectors module. The motion of the swinging platform was defined using the Rotary Motion pair in the Motions module. In this experiment, the pivot point for the swinging platform's motion was located 180 mm below the center of the platform's surface.

The motion of the platform is a sinusoidal excitation motion with a fixed amplitude relative to the pivot point. The drive functions for each degree of freedom motion are defined as follows:

$$A \times \sin(2\pi/T \times t)$$

where A represents the motion amplitude, which, in the case of oscillatory motion, signifies the swing amplitude in radians, and in the case of pendulum motion, it represents the displacement amplitude; T is the motion period; t is the time variable.

3.2.4. Time Step Setting

In the simulation calculations, the time step size has a significant impact on the stability and accuracy of the results. A shorter time step results in higher accuracy. Figure 14 shows the support force curve of the pedestal calculated by setting the analysis steps at 10, 20, 50, 100, 500 and 800 steps per second under the M3 condition. When the analysis step is below 100 steps per second, there are a lot of unreasonable fluctuations in the curve. When the analysis step is greater than or equal to 100 steps per second, the resulting calculation curve remains smooth.

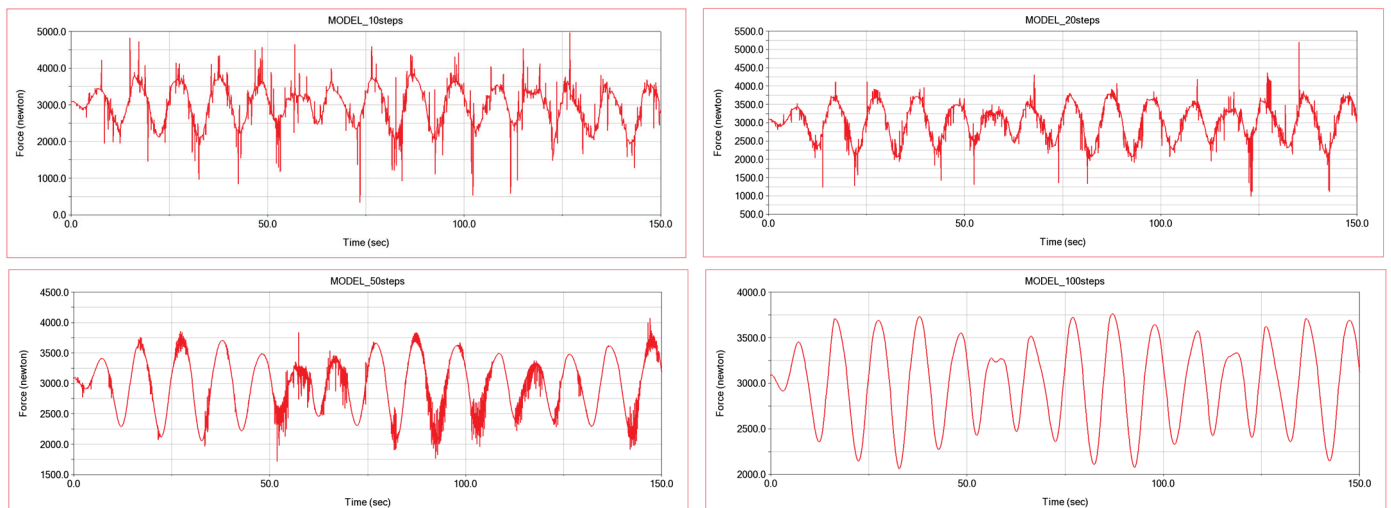


Figure 14. Cont.

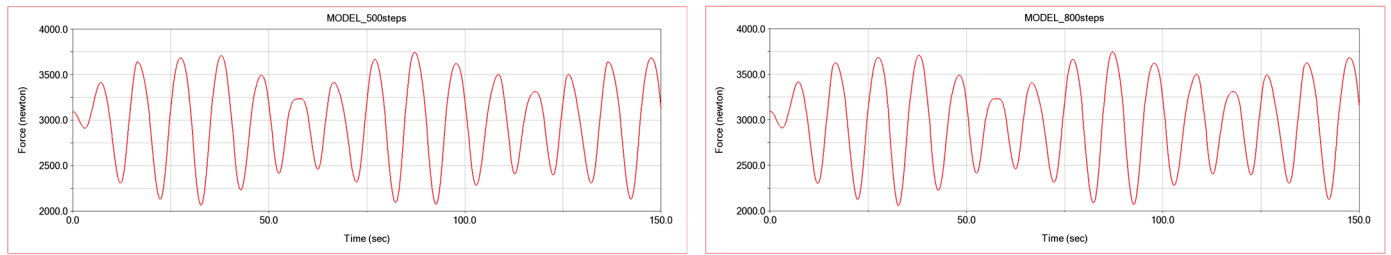


Figure 14. Comparison of calculation results from different analysis steps.

In order to take into account both calculation accuracy and calculation efficiency, in this simulation, the time step was set to 500 steps per second.

4. Results and Discussion

4.1. Support Force of Pedestal

When comparing the pedestal support force curves for each working condition between the test and simulation, it can be observed that the overall trends are consistent, except for some differences in the peak load. Comparing the calculated conditions for different sets of motion parameters, it can be seen that, as the motion amplitude increases, the magnitude of the pedestal support force variation also increases accordingly. Furthermore, in the eccentric placement condition, where the pedestals are arranged toward the rear of the model, the front pedestals are closer to the model's center of gravity, resulting in higher support forces compared to the rear pedestals and the pedestals in the centered condition. When designing similar securing schemes, it is advisable to avoid situations where support structures are biased toward one end to reduce the likelihood of load concentration on a specific support component. See Figure 15.

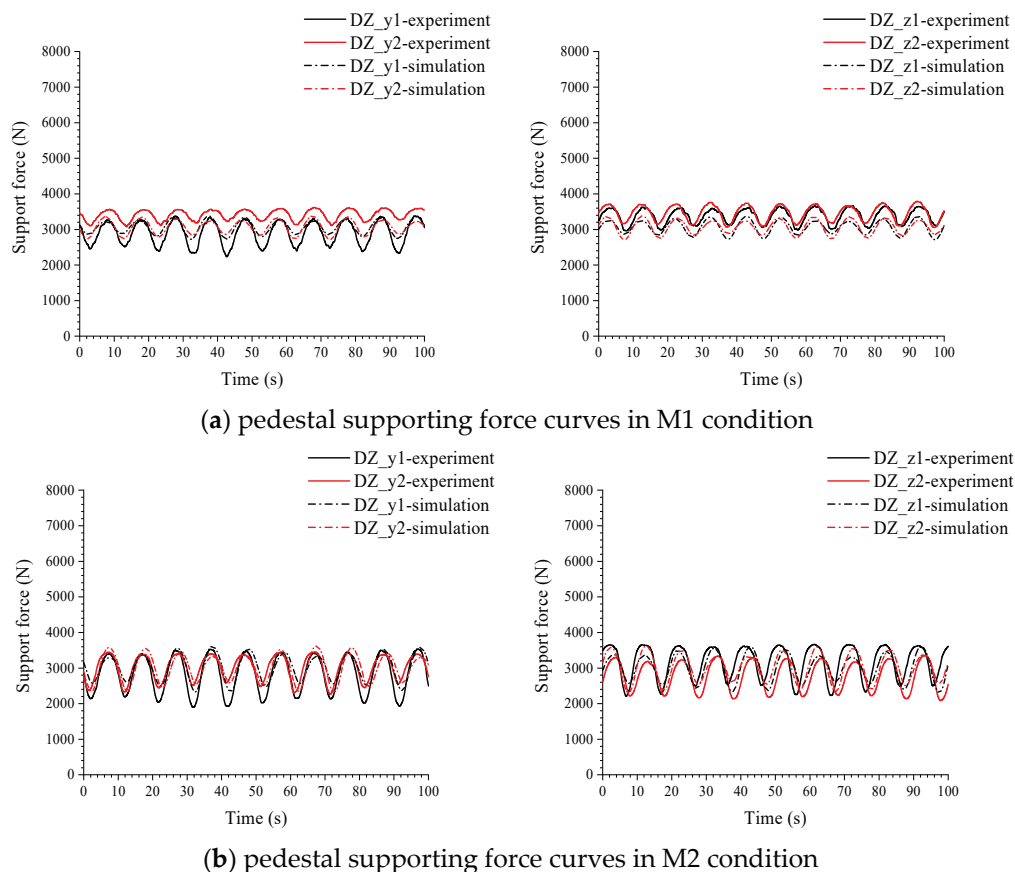
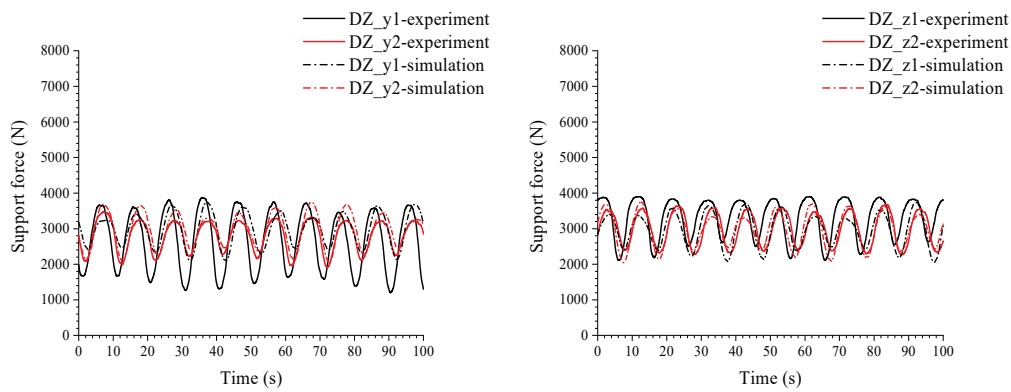
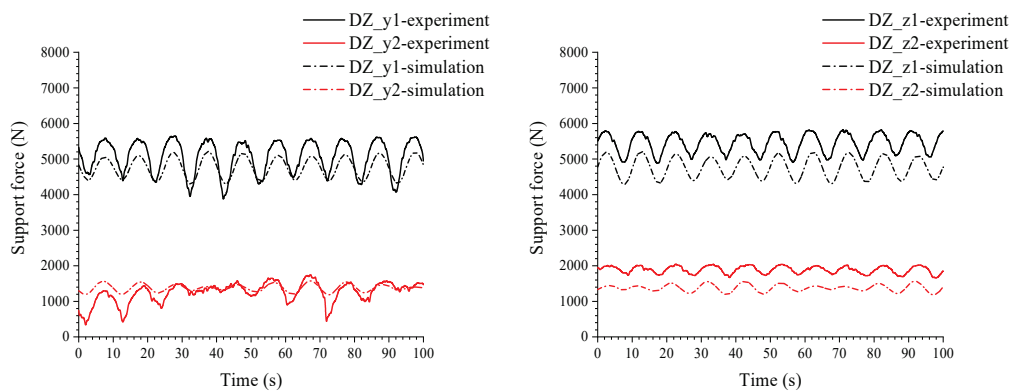


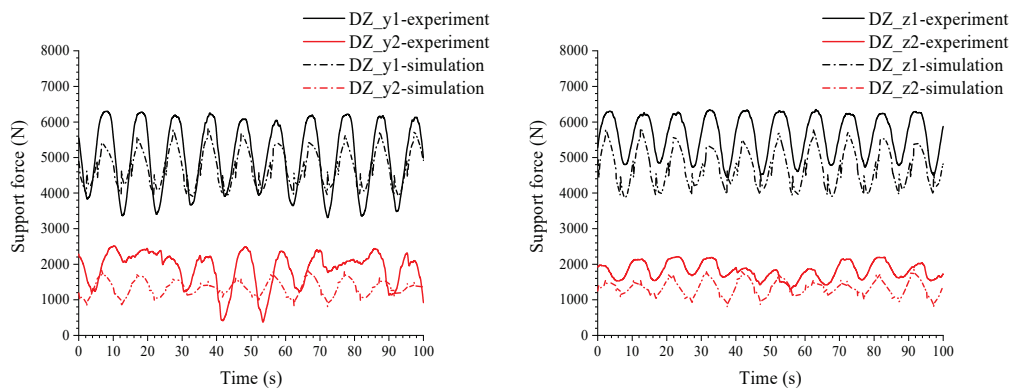
Figure 15. Cont.



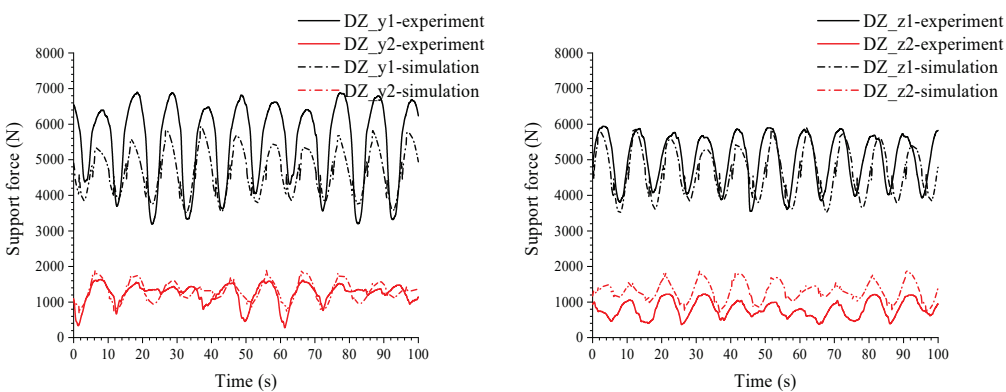
(c) pedestal supporting force curves in M3 condition



(d) pedestal supporting force curves in O1 condition



(e) pedestal supporting force curves in O2 condition



(f) pedestal supporting force curves in O3 condition

Figure 15. Comparison of supporting force curves of pedestal under different conditions.

4.2. Lashing Force

The comparison between test and simulation for the rope loads is similar to that of the pedestal support forces. The overall trends are consistent, except for some differences in the peak load. As the motion amplitude increases, the magnitude of the rope load variation also increases accordingly. In the offset condition, the ropes that are under tension in the model's suspended section experience larger load variations compared to the other ropes. This is attributed to the greater relative displacement of the suspended section during motion, resulting in substantial changes in rope lengths. In all working conditions, the rope loads exhibit periodic variations within each motion cycle. This periodicity is due to differences in the periods of motion for different degrees of freedom, resulting in various phase differences during the motion process. Therefore, when verifying the safety of securing schemes, it is essential to consider the motion states with different phase differences to ensure the accuracy of the verification results. There is sudden fluctuation in the partial load curve of some test conditions, but it does not appear in the corresponding simulation conditions. It is presumed to be caused by local vibration during the test. See Figures 16–21.

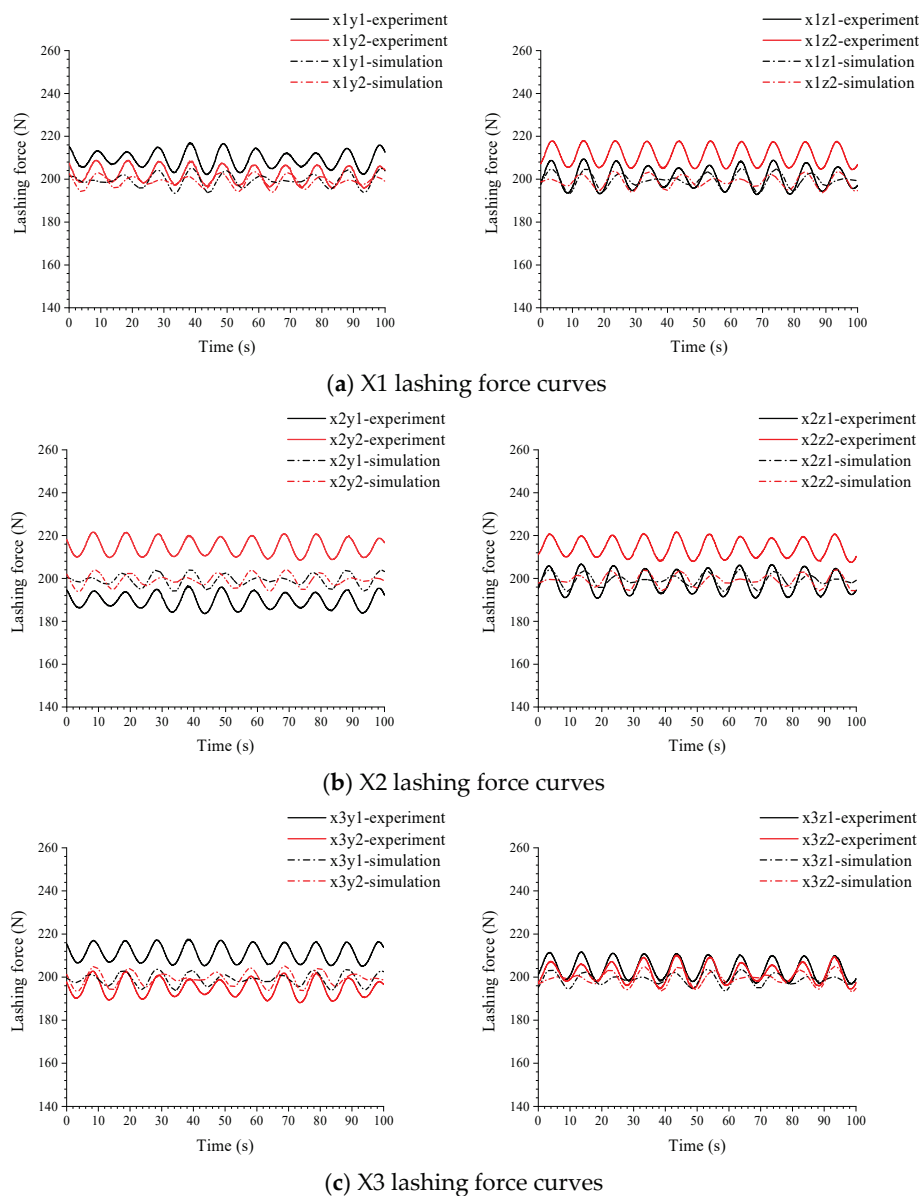
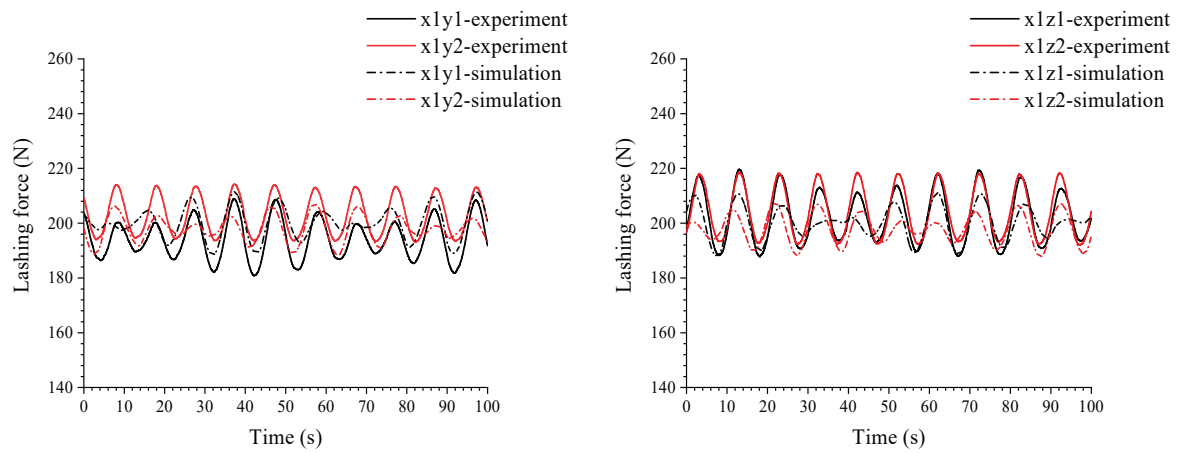
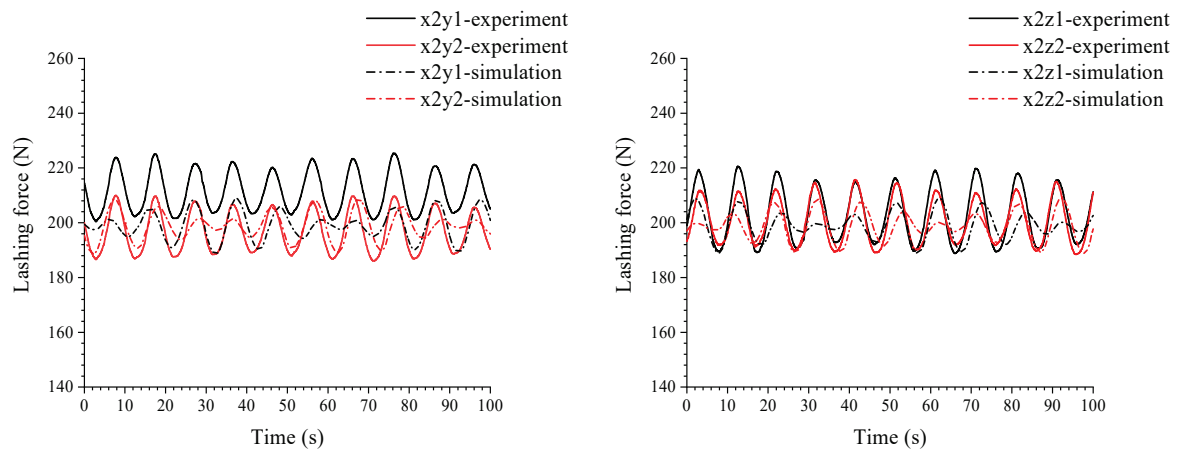


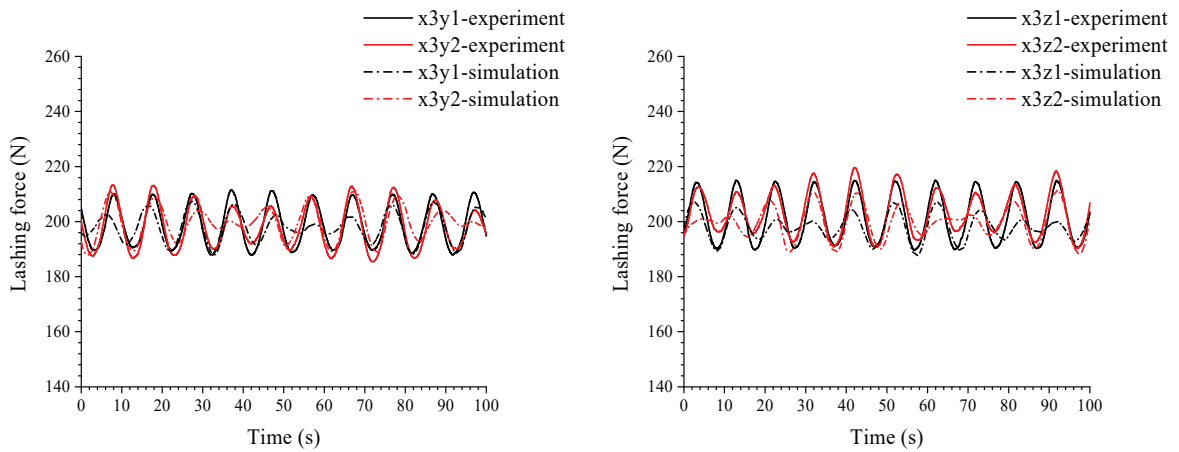
Figure 16. Comparison of lashing force curves of each rope in M1 condition.



(a) X1 lashing force curves

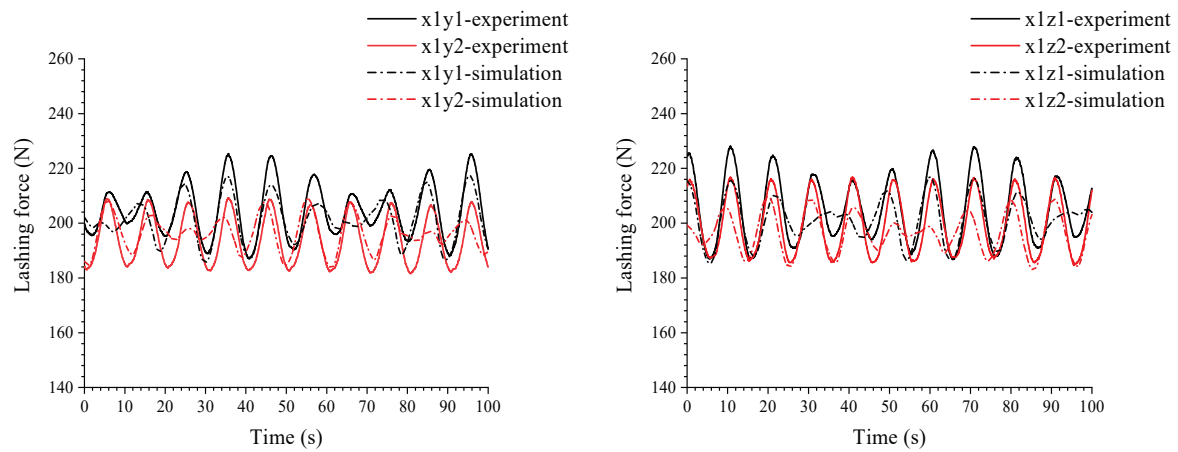


(b) X2 lashing force curves

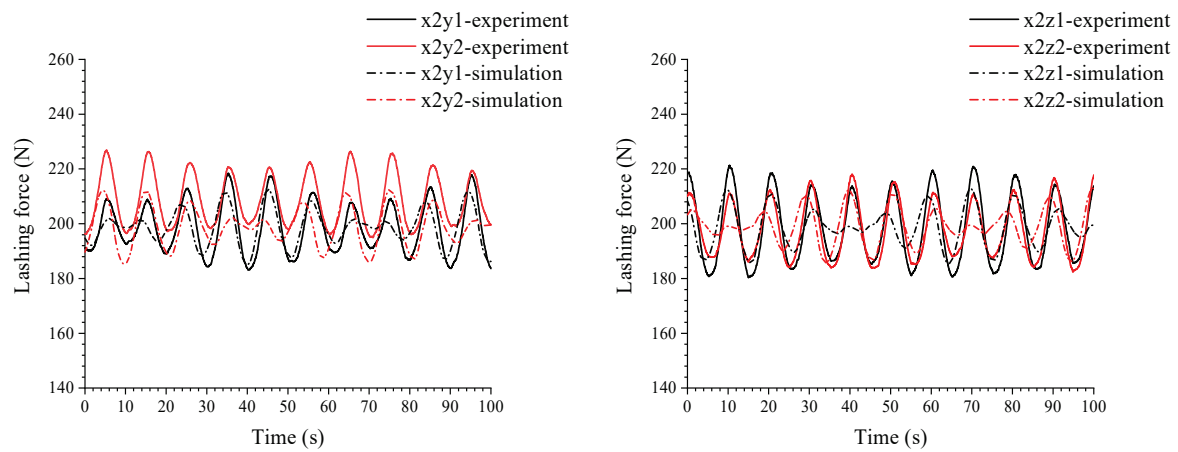


(c) X3 lashing force curves

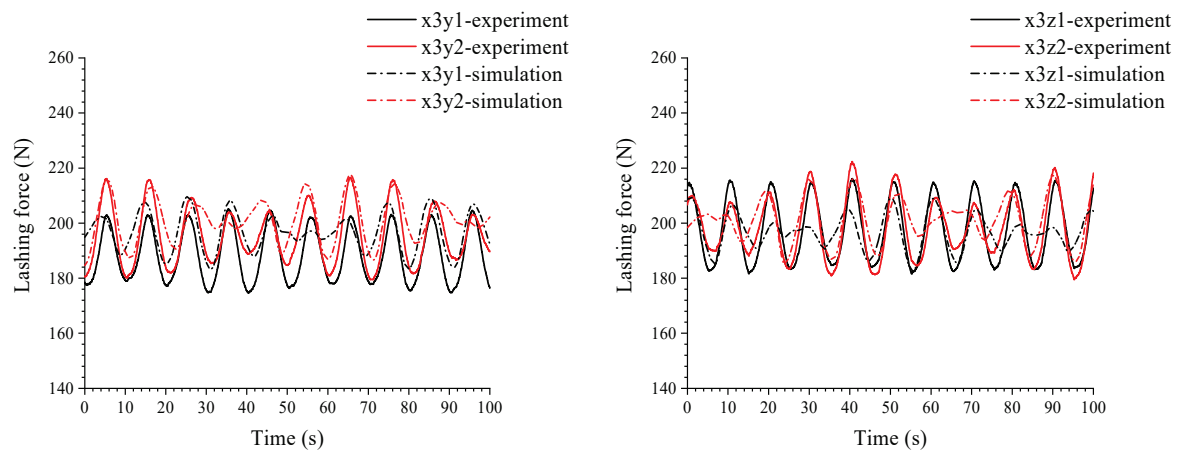
Figure 17. Comparison of lashing force curves of each rope in M2 condition.



(a) X1 lashing force curves



(b) X2 lashing force curves



(c) X3 lashing force curves

Figure 18. Comparison of lashing force curves of each rope in M3 condition.

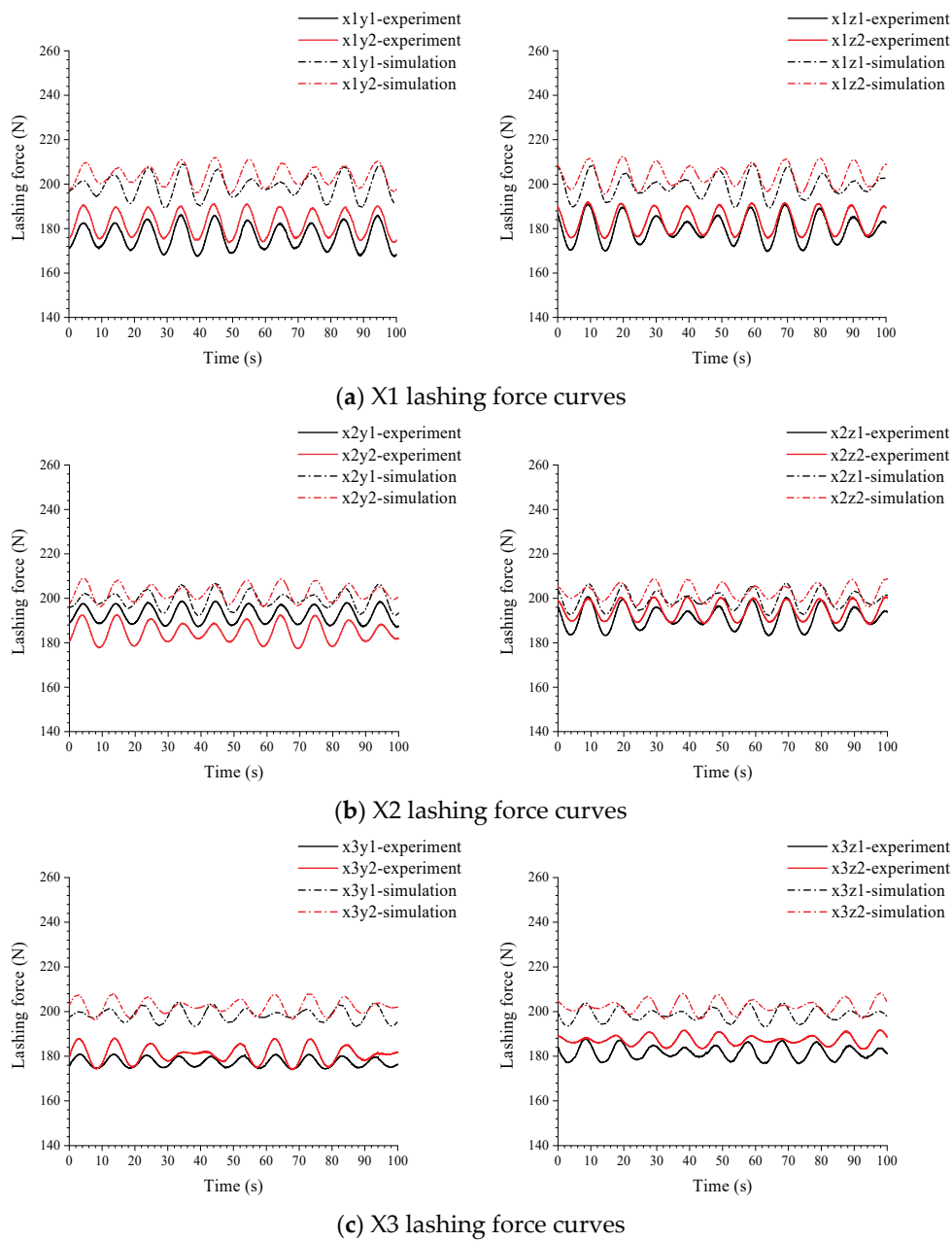


Figure 19. Comparison of lashing force curves of each rope in O1 condition.

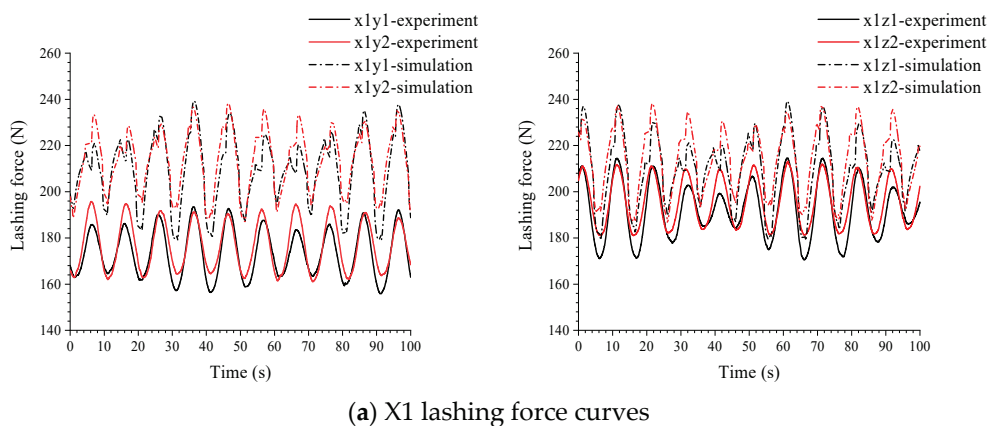
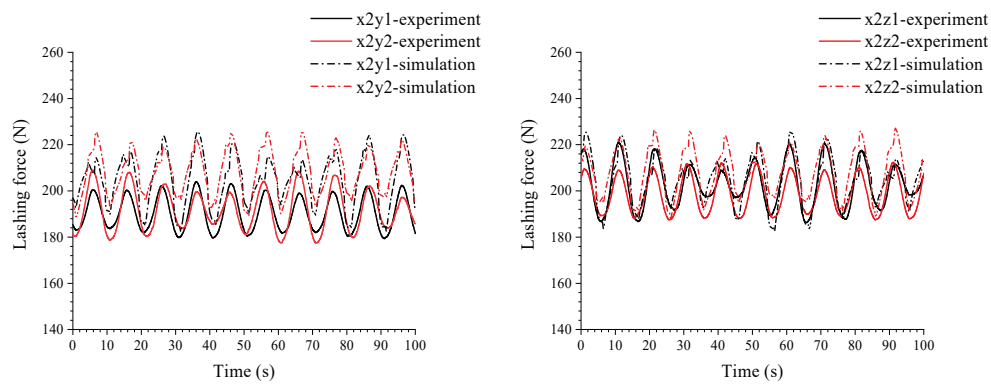
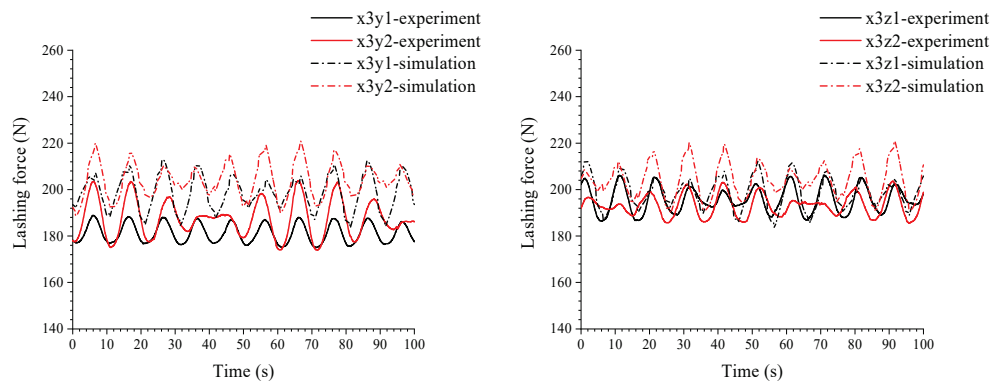


Figure 20. Cont.

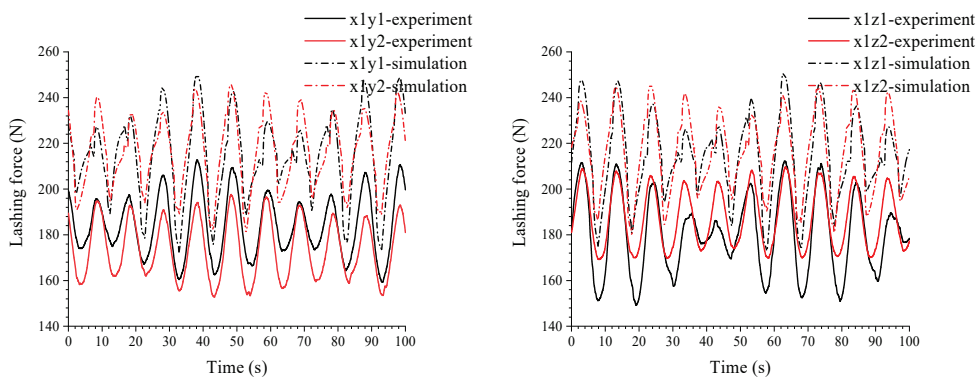


(b) X2 lashing force curves

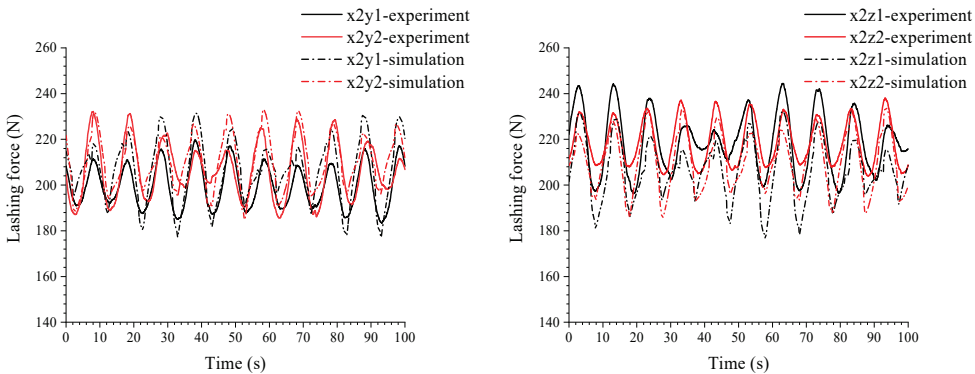


(c) X3 lashing force curves

Figure 20. Comparison of lashing force curves of each rope in O2 condition.

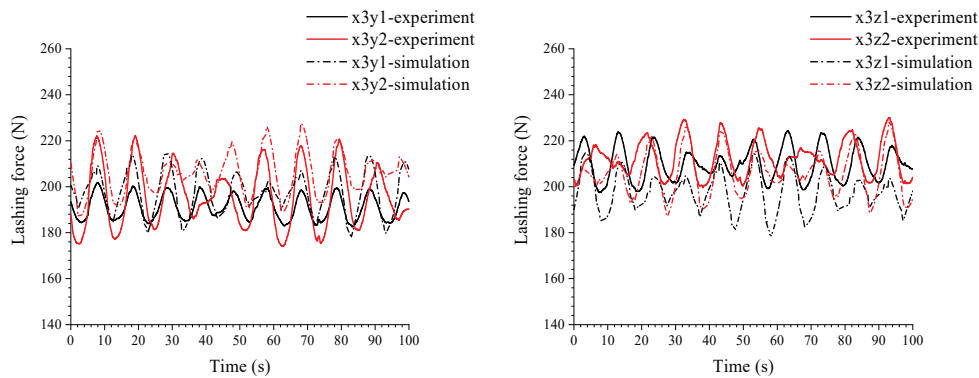


(a) X1 lashing force curves



(b) X2 lashing force curves

Figure 21. Cont.



(c) X3 lashing force curves

Figure 21. Comparison of lashing force curves of each rope in O3 condition.

The smaller amplitude of rope load variations compared to the 200 N pre-tension force is attributed to the support provided by the pedestal. The support structure of the pedestal directly restricts the model's displacements in various directions other than the vertical upward direction during motion. In the test, the model's motion does not generate sufficient acceleration to cause the model to detach from the pedestal. As a result, the relative displacements induced by the model's motion are relatively small, leading to minimal deformation effects on the ropes.

4.3. Load Peak and Error Analysis

In previous studies, the safety of the securing scheme was mainly verified by checking whether the load of each component exceeded the allowable load. In this study, the error of the peak load of each component is used as the criterion for the accuracy of the simulation method. The peak load of each component and the error between the test and simulation results are summarized in Table 3.

Table 3. Summary of load peak values for each component under different conditions.

Condition ID	Support Force (N)			Lashing Force (N)		
	Test	Simulation	Error	Test	Simulation	Error
M1	3738	3380.3	9.57%	221.7	204.7	7.67%
M2	3661.1	3601.1	1.64%	225.1	211.1	6.22%
M3	3900.2	3744.6	3.99%	228.2	216.8	5.00%
O1	5765	5214.5	9.55%	200.4	212.2	5.89%
O2	6331.4	5815.2	8.15%	219.4	238.8	8.84%
O3	6865.1	5881.9	14.32%	244.2	250.53	2.59%

Pedestal and rope loads:

For pedestal load peaks, the error is relatively large, particularly in the O3 condition, where the error is around 15%. However, in most cases, the error between test and simulation load peaks is within 10%. The errors in pedestal loads are generally acceptable. The O3 condition, with a slightly larger error, might indicate specific challenges or conditions not adequately captured by the simulation. The error in rope load peaks is generally within 9%.

The data from the simulation in ADAMS agrees reasonably with the experimental data, indicating that numerical simulations conducted using ADAMS can provide valuable reference for the verification and optimization of securing schemes.

Sources of error:

- Numerical simplifications

In numerical simulations, simplifications were made for the test model, pedestal supports, and ropes by assuming that all components are rigid structures and neglecting

the local deformation effects of the structures. Additionally, in the numerical model, springs were used to simulate the chain, leading to the introduction of some approximations.

- Misalignments and experimental imperfections

The actual experimental setup is not as perfect as in the numerical simulations, and there is a lack of alignment between the support piers and the cylinder. These differences between the simplified numerical models and the actual experimental conditions can lead to discrepancies in the results.

- Pre-tensioning forces

The application of pre-tensioning forces to the ropes is achieved by adjusting the basket bolts. In practical operations, the mutual influence between multiple pre-tensioning adjustments and manual operation can make it challenging to simultaneously achieve the predetermined pre-tensioning forces for all ropes. During the motion of the swing test platform, there may also be gradual relaxation of the pre-tensioning forces. These factors can introduce discrepancies compared to the numerical model, which assumes precise relative positions and accurate pre-tensioning forces.

5. Conclusions

Based on a self-designed securing scheme experiment, data for lashing loads and pedestal support forces in the test scheme were collected. Variations of various loads under different securing methods and platform movements were analyzed. By setting different motion parameters and equipment placement conditions, the study has analyzed the time–history curves of the support forces on the support pedestals and the rope loads under the influence of different factors. The patterns of load changes for various components are revealed when considering motion states and securing configurations. Based on these findings, a numerical simulation was conducted. By comparing the simulation results with the experimental results, the reliability of the simulation method was verified. The results show that the simulation method used in this study can be used as a reference for the safety check of the securing scheme. The main conclusions of this study are as follows:

1. When comparing the experimental data with the numerical simulation results for various operating conditions, the periodicity, and trends of the support forces on the pedestal supports and rope loads are consistent, with only minor discrepancies in peak load values. Notably, the support forces exhibit relatively larger discrepancies compared to the rope loads. For most cases, the peak load discrepancies between experiments and simulations are within 10%, except for a few specific conditions where they reach around 15%. The discrepancies in peak rope loads are generally within 9%. This demonstrates that the simulation method can be applied effectively for engineering analysis and prediction purposes.
2. With the increase in the amplitude of the designed motion parameters, the response amplitudes of various load components increase in all test conditions. The support forces on the pedestal exhibit more pronounced variations, while the rope loads show relatively smaller variations. This is primarily because the support configuration of the pedestal supports can directly restrict the model's displacement in directions other than vertical during the motion. Consequently, in the test conditions, the model's motion does not generate significant relative displacement, resulting in minimal deformation effects on the cables.
3. In the offset condition, the pedestal support force of the forward pedestal is significantly greater than that in the centered condition. This is due to the fact that, in the offset condition, the model's center of gravity is biased toward the forward pedestal, resulting in a larger load being borne by the forward pedestal. In practical structural design of securing systems, it is advisable to avoid situations where the supporting structures are biased towards one end in order to reduce the likelihood of load concentration on a particular support component.

4. The discrepancies between numerical simulations and model experiments in the loads on structural components may arise from two main factors. Firstly, the simulation model assumes a rigid body model, neglecting local deformation effects. Secondly, during the experimental implementation, the arrangement of various components may not be as perfect as in numerical simulations. The pre-tension forces applied to the ropes at different locations can influence each other, and it is challenging to simultaneously achieve their predetermined values through manual adjustments.

The use of simulation technology can provide a more comprehensive reference for the safety check of the securing scheme. However, due to technical limitations, the simulation method of securing rigging in this study can be further improved. In addition, because the assembly method poses difficulty in achieving accuracy of the simulation setting, the error caused by the assembly method can also be further studied.

Author Contributions: Formal analysis, M.L.; methodology, M.L.; project administration, G.W. and K.L.; resources, K.L.; supervision, K.L. and J.W.; validation, M.L.; writing—original draft, M.L.; writing—review and editing, Y.L. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by Project supported by the National Natural Science Foundation of China grant number 52171311, and Open Project of the National Key Laboratory of Waterway Traffic Control grant number W23CG090243.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Choung, J.; Jeong, S.-M. Vehicle Securing Safety Assessments of a Korean Coastal Car Ferry According to Acceleration Prediction Approaches. *Brodogradnja* **2019**, *70*, 115–131. [CrossRef]
2. Acanfora, M.; Montewka, J.; Hinz, T.; Matusiak, J. On the Estimation of the Design Loads on Container Stacks Due to Excessive Acceleration in Adverse Weather Conditions. *Mar. Struct.* **2017**, *53*, 105–123. [CrossRef]
3. Kabaciński, J.; Wiśnicki, B. Accuracy Analysis of Stowing Computations for Securing Non-Standard Cargoes on Ships According to IMO CSS Code. *Pol. Marit. Res.* **2009**, *16*, 67–71. [CrossRef]
4. Pérez-Canosa, J.M.; Orosa, J.A.; de Osés, F.X.M.; Lama-Carballo, F.J. Optimization of Ship's Navigational Parameters to Improve the Stowage and Securing Criteria of Non-Standardized Cargo in Ships. *J. Mar. Sci. Eng.* **2023**, *11*, 1782. [CrossRef]
5. Sternsson, M.; Björkenstam, U. Statistical Study of Lashing Forces Measured On-Board a PCTC Vessel. *Mar. Struct.* **2003**, *16*, 345–354. [CrossRef]
6. Jia, J.; Ulfvarson, A. Dynamic Analysis of Vehicle—Deck Interactions. *Ocean Eng.* **2006**, *33*, 1765–1795. [CrossRef]
7. Jia, J. Investigations of Vehicle Securing without Lashings for Ro-Ro Ships. *J. Mar. Sci. Technol.* **2007**, *12*, 43–57. [CrossRef]
8. Matusiak, J. Dynamics of Cargo Shift Onboard a Ship in Irregular Beam Waves. *Int. Shipbuild. Prog.* **2000**, *47*, 77–93.
9. Andersson, P.; Jagelčák, J.; Lind, E.; Petersen, S.S. *Report from Pulling Tests with Used Lashing Equipment*; MariTerm AB: Höganäs, Sweden, 2009.
10. Turnbull, S.R.; Dawson, D. The Securing of Rigid Semi-Trailers on Roll-on/Roll-off Ships. *Int. J. Mech. Sci.* **1997**, *39*, 1–14. [CrossRef]
11. Surendran, S.; Lee, S.K.; Reddy, J.V.R.; Lee, G. Non-Linear Roll Dynamics of a Ro-Ro Ship in Waves. *Ocean Eng.* **2005**, *32*, 1818–1828. [CrossRef]
12. Kawamura, K.; Hashimoto, H.; Matsuda, A.; Terada, D. SPH Simulation of Ship Behaviour in Severe Water-Shipping Situations. *Ocean Eng.* **2016**, *120*, 220–229. [CrossRef]
13. Liu, J.; Li, C.; Wang, D.; Cai, Z. Investigations on the Dynamics of Container Stack and Securing System under Rolling Motion Using a Scaled Model Test. *Ships Offshore Struct.* **2022**, *17*, 92–104. [CrossRef]
14. Li, C.; Wang, D. Multi-Objective Optimisation of a Container Ship Lashing Bridge Using Knowledge-Based Engineering. *Ships Offshore Struct.* **2019**, *14*, 35–52. [CrossRef]
15. Li, C.; Wang, D.; Liu, J. Numerical Analysis and Experimental Study on the Scaled Model of a Container Ship Lashing Bridge. *Ocean Eng.* **2020**, *201*, 107095. [CrossRef]
16. Li, C.; Wang, D.; Liu, J.; Cai, Z. Experimental and Numerical Investigation on Dynamic Response of a Four-Tier Container Stack and Lashing System Subject to Rolling and Pitching Excitation. *Appl. Ocean Res.* **2021**, *109*, 102553. [CrossRef]

17. Song, F.; He, Y.; Liu, H. Research on Goods and the Ship Interaction Based on ADAMS. *MATEC Web Conf.* **2017**, *95*, 11002. [CrossRef]
18. Kreuzer, E.; Schlegel, V.; Stache, F. Multibody Simulation Tool for the Calculation of Lashing Loads on RoRo Ships. *Multibody Syst. Dyn.* **2007**, *18*, 73–80. [CrossRef]
19. Nam, W.; Park, S.-J.; Kim, K. Numerical Investigation of the Ultimate Strength of D-Ring Devices and Deck Structures. *J. Mar. Sci. Eng.* **2022**, *10*, 952. [CrossRef]
20. Zhu, H.J.; Yao, H. Dynamic Simulation Analysis of a Ship-Borne Loading Device. *Adv. Mater. Res.* **2012**, *487*, 562–567. [CrossRef]
21. E-Chao, L.; Bai-Jun, T.; Yang-Gui, L. Amendments to the Methods of Non-Standardized Cargo Unit Securing Assessing. In Proceedings of the 2020 IEEE 5th International Conference on Intelligent Transportation Engineering (ICITE), Beijing, China, 11–13 September 2020; IEEE: Piscataway, NJ, USA, 2020; pp. 371–375. [CrossRef]
22. Adams, G.G.; Nosonovsky, M. Contact Modeling—Forces. *Tribol. Int.* **2000**, *33*, 431–442. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Dynamic Stability Analysis of Subsea Tunnel Crossing Active Fault Zone: A Case Study

Zhiqiang Li ^{1,2}, Qiushi Liu ², Shian Liu ^{1,3}, Xueying Liu ^{1,3}, Yongqiang Zhang ^{1,3}, Shixiang Jia ^{1,3},
Guoliang Song ^{1,3} and Yuting Zhao ^{1,3,*}

- ¹ Key Laboratory of Geological Safety of Coastal Urban Underground Space, Ministry of Natural Resources, Qingdao 266061, China; zhiqiangli@sdu.edu.cn (Z.L.); 13780608551@163.com (S.L.); liuxueying@hnu.edu.cn (X.L.); sam_mhl@163.com (Y.Z.); 13791835815@163.com (S.J.); 13589269012@163.com (G.S.)
- ² Geotechnical and Structural Engineering Research Center, Shandong University, Jinan 250061, China; 202114983@mail.sdu.edu.cn
- ³ Qingdao Geo-Engineering Surveying Institute (Qingdao Geological Exploration Development Bureau), Qingdao 266061, China
- * Correspondence: ytz_2020@163.com

Abstract: The rock strength in an active fault zone is low and the surrounding rock is fractured and has poor stability, making any subsea tunnel crossing the active fault zone extremely susceptible to disasters such as tunnel collapse, sudden water ingress, and mud inrush. This poses a potential threat to the construction project, making the dynamic stability analysis of a subsea tunnel crossing an active fault zone of great significance. This study takes the second subsea tunnel crossing the Cangkou Fault in Jiaozhou Bay as the engineering background and conducts numerical simulations by employing different lining stiffnesses for tunnel excavation, as well as applying dynamic loads. The dynamic stability of the subsea tunnel crossing the active fault zone is evaluated by comparing and analyzing the lining's displacement, peak acceleration, and stress characteristics. This study explores the disaster-causing mechanisms of active fractures, determining that the hazard of orthogonal misalignment in an active fault zone is the least severe, while the hazard of opposite misalignment is the most severe. This research provides a basis for disaster prevention and mitigation in active fracture zones.

Keywords: subsea tunnel; numerical simulation; dynamic stability; active fault zone

1. Introduction

Subsea tunnels are often located at the edges of continental frames and islands where seismic activity is intense, crossing various geological strata and sometimes intersecting with active fault zones. When the construction of tunnels inevitably crosses active fault zones, the activity patterns and deformation characteristics of these zones during the effective period of the project will affect the safety and stability of the engineering work [1,2]. Potential earthquakes and misalignments of fracture zones can lead to the opening and closing of rock mass structural surfaces, slippage, and the initiation–expansion–coalescence of cracks at these active fault zones crossed by subsea tunnels. This reduces the stability of the rock mass and, in severe cases, can even cause structure damage and deformation to the tunnel, destroy the tunnel's waterproofing facilities, and eventually trigger a chain of secondary geological disasters, such as large deformations, collapses, sudden water ingress, and mud inrush, posing a serious threat to the long-term safety and maintenance of subsea tunnels [3,4].

Due to the principle of avoiding active faults in tunnel engineering design, there are few engineering cases of tunnels crossing active fault zones. The characteristics of internal force distribution, deformation patterns, and failure modes within the rock mass of fault

zones under fault displacement and seismic actions are closely related to the dynamic response of submarine tunnels to fault zones [5–7]. The seismic damage to fault-crossing tunnels is considered to have two main causes: one is the vibrational damage to the tunnel caused by earthquakes; the other is the shear damage to the tunnel caused by the fault. When the engineering geological conditions where the tunnel is located are good, the tunnel itself has better seismic performance. Tunnel seismic damage generally occurs more frequently in areas with complex geological conditions, such as poor rock mass quality and significant variations in stratum conditions. He et al. [8] found that tunnels crossing fault zones are prone to damage in earthquakes, and the fault zone sections are the controlling areas for the tunnel's resistance to seismic damage. The main forms of seismic damage are displacement and various types of cracking. Currently, the main theoretical analysis method for tunnels' seismic resistance is the analytical method, with the pseudo-static method being the most widely used within the analytical methods. However, due to constraints that differ from above-ground structures, the results of this method tend to be conservative. Research on the dynamic response and stability of tunnels can also be conducted through model experiments and numerical simulations. Fang et al. [9], based on a shaking table, studied the dynamic response patterns and characteristics of fault-crossing tunnels under different vibration directions. They found that faults could weaken the propagation of seismic waves in the rock mass to a certain extent. Shahidi et al. [10] studied the longitudinal seismic dynamic response characteristics of the Koohrang-III water conveyance tunnel in Iran through numerical simulation.

The impact of active fault zones on submarine tunnels mainly has three aspects: one is earthquakes induced by sudden stick-slip deformation, the second is the effect of long-term creep deformation on the structure, and the third is the impact of distant earthquakes on active fault zones [6,7,11,12]. Among these, the creep deformation of the fault zone under tectonic stress is long-term, and it is necessary to design corresponding tunnel structures to prevent its harm to the tunnel structure in advance. Sudden stick-slip deformation, characterized by large displacement, a high displacement gradient, and the potential to induce high-magnitude earthquakes, is mainly controlled by limiting the range of damage in engineering projects. In response to the impact of active fault displacement on tunnel structures, Kiani et al. [13] proposed a new method for assessing tunnel vulnerability curves based on centrifuge experiments and studied the deformation and force patterns of fault-crossing tunnels under the action of normal faults. Ma et al. [5] revealed the damage mechanisms of tunnels under the displacement effect of strike-slip and thrust faults through numerical simulation, from aspects such as lining deformation, stress distribution, and the distribution of plastic zones, finding that lining damage is significantly influenced by the displacement of active faults. Liu et al. [14], based on model experiments, studied the effect of fault dip angle on the mechanical response characteristics of tunnel structures, discovering that as the fault dip angle decreases, the tunnel structure's mode of failure shifts from bending to shear damage. Due to the complex force characteristics of submarine tunnels in complex seabed environments under seismic actions, their damage mechanisms are diversified under the effect of earthquakes. Influenced by the interaction between marine engineering geology and tunnel structures, as well as the stability of marine engineering geological bodies, the issue of seismic disasters becomes even more complicated [15–17].

In this study, a series of numerical simulations are conducted to analyze the influence of the dynamic seismic action and the motion of active fault zones on the dynamic stability of a subsea tunnel crossing an active fault zone. Section 2 briefly introduces the engineering background of the second subsea tunnel in Jiaozhou Bay, Qingdao. Section 3 shows the detailed numerical simulation and study of the dynamic stability of the subsea tunnel under different support conditions. Section 4 shows the results of the numerical simulation of the dynamic stability of the subsea tunnel under seismic effects and fault motion. Section 5 is the discussion and limitations section of this paper. The conclusions are given in Section 6.

2. Engineering Background

The second subsea tunnel in Jiaozhou Bay will connect Qingdao and Huangdao and be a transportation corridor and critical infrastructure asset linking various blue economic zones. With a total length of 14.36 km, it will be Asia's longest subsea road tunnel and the world's most giant subsea concealed dredge road tunnel [18].

The coast of Jiaozhou Bay has typical mountainous bedrock, and the nearby topography and geomorphology are influenced by rock lithology and geological structure, and external forces influence the geomorphology. On the whole, Jiaozhou Bay has a large belly and small mouth, deep water in the southeast, sloping topography from north to south, and a flat seabed in the north of the bay. According to the preliminary survey results, the stratum of the Jiaozhou Bay area is mainly granite, with local development of gabbro veins, and the surface layer is covered with a small amount of residual soil and a weathering layer. The geotechnical type is relatively singular and uniform, except for the possible development of a local tectonic fracture zone. There is no other particular geotechnical distribution. The area where the project is located crosses two first-class geotectonic units of North China, and the Yangzi fault block, and the project line is basically located in the area of the North China fault block where the seismic structure is developed. Among them, the north–northeast-oriented fractures are large in scale and have a long history of activity, while the northwest-oriented fractures show a Neozoic nature and control the Late Cenozoic basin. The main source of earthquakes in the project area is the strong seismic activity of the Tanlu Fault Zone and Bohai–Penglai Fault Zone. Combined with the regional geological background, the near-field area of the project has the conditions for earthquakes, and earthquakes of magnitude 6 to 7 may occur within one hundred years.

The second subsea tunnel in Jiaozhou Bay crosses the Cangkou Fault (F52 in Figure 1), which has more fractured rocks and develops several cross-sections. During the construction process, many disaster problems may emerge, such as issues causing local relative displacement of the tunnel and changing the force and surrounding rock stress of the tunnel structure, which will cause incalculable damage to the subsea tunnel.

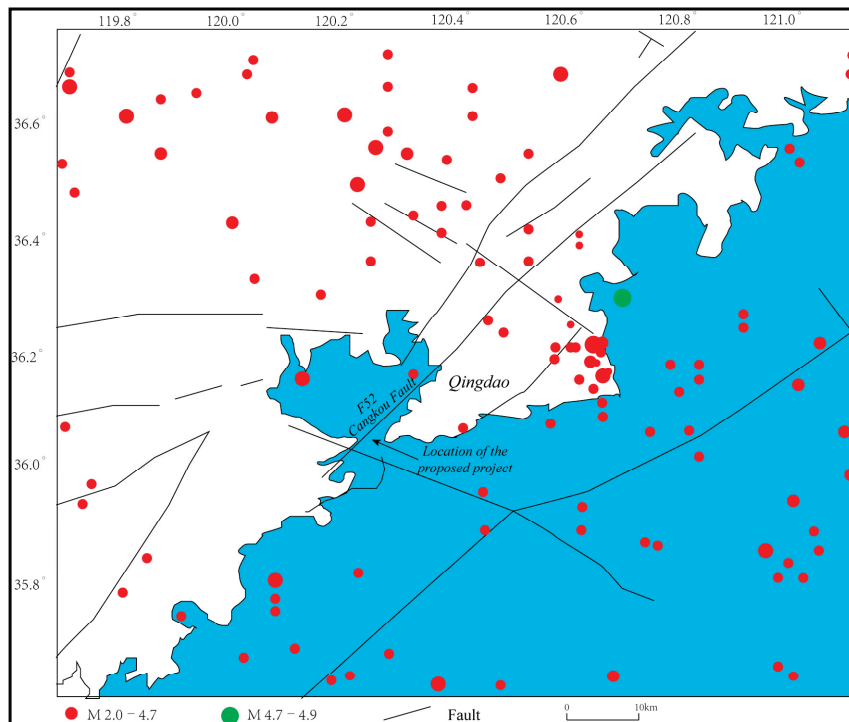


Figure 1. Distribution of epicenter and fault zone in Qingdao and its adjacent areas.

3. Numerical Simulation

3.1. Model Geometry

This study uses the second subsea tunnel in Jiaozhou Bay, crossing the Cangkou Fault, as an example. The first consideration is to build a model of a suitable size. The mesh division needs to strike a balance between computational cost and accuracy. Excessive density incurs higher computational times, while insufficient density compromises accuracy. Balancing comprehensive site conditions and model simplification, this paper determines the subsea tunnel model, for the left and right directions, should extend 40 m on either side. The upward direction, or buried depth, is set at 45 m, downward as 25 m, and along the tunnel axis as 120 m. The tunnel radius is 6.5 m. After dividing the grid, there are 109,739 cells and 116,684 nodes. The rock mass in the numerical simulation is modeled using the Mohr–Coulomb model, and the excavated section is set as a null model.

The model (shown in Figure 2) is divided into eight groups. The active fault zone is 10 m thick, with an inclination of 62.9°, and is located between 20–65 m in the Y-direction. The overburdened miscellaneous fill is 15 m thick. The blue part represents the hanging wall of the active fault zone, and the brown part represents the footwall of the active fault zone. The blue and brown areas are slightly weathered granite, divided into two by the active fault zone. The tunnel inside the middle of the model represents the tunnel. The initial support, the secondary lining, and the grouting reinforcement are set around the tunnel one by one.

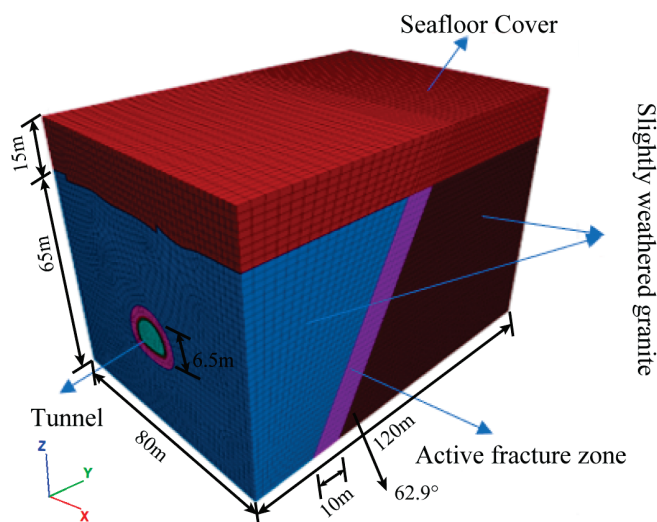


Figure 2. Schematic diagram of the model.

To control the accuracy and make the dynamic calculation more clear and definite, the cell size must meet specific requirements. For seismic waves to be accurately described in the model, the dimensions of the dividing cell must satisfy:

$$\Delta l \leq \frac{\lambda}{10} \sim \frac{\lambda}{8} \quad (1)$$

where λ is the wavelength corresponding to the highest frequency.

The allowable cell size is 5.3 m, while the maximum cell size in this model is 3.2 m, which meets the accuracy requirement.

Along the tunnel, a total of nine typical cross-sections are axially selected at 10 m, 20 m, 30 m, 40 m (within the fault), 50 m, 60 m, 70 m, 90 m, and 110 m for the placement of monitoring points. For the layout of these monitoring points, four are placed in each tunnel section at the top of the arch, arch waist, arch foot, and bottom of the invert, respectively. The monitoring points' layout and specific locations are shown in Figure 3.

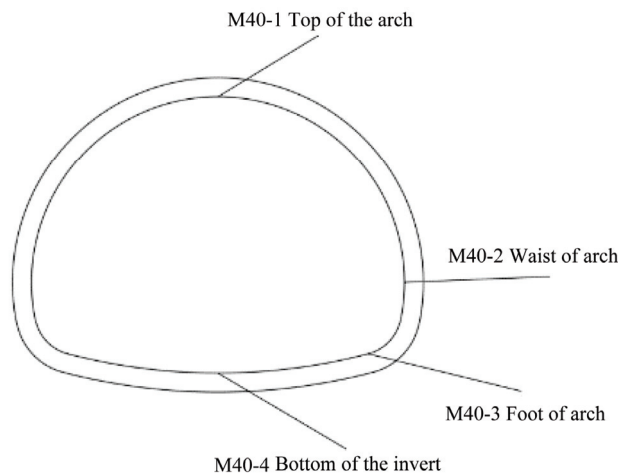


Figure 3. Monitoring point arrangement for the tunnel section at Y = 40 m.

3.2. Material Parameter

For the parameters of geotechnical units, based on the existing engineering cases and the geological data of Jiaozhou Bay, Qingdao, the following necessary stratigraphic mechanical parameters are determined, as shown in Table 1.

Table 1. Mechanical parameters of the stratum.

Material	Volumetric Mass/kg/m ³	Bulk Modulus/MPa	Shear Modulus/MPa	Cohesion/kPa	Internal Friction Angle/°	Strength of Tension/kPa
Seafloor cover	1900	17.9	1.86	5	18.0	0
Active fault zone	2200	18	4530	300	45	100
Slightly weathered granite	2720	7550	5660	6050	58	4257

Three contact surfaces are set before, after, and at the top of the active fault zone, and two more contact surfaces are set to separate the overlying soil and rock layers. The parameters of the contact face are that the tangential stiffness is 4.66×10^{10} N/m, the cohesion is 400 kPa, and the internal friction angle is 24° .

3.3. Boundary Conditions

The displacement boundary conditions of the model in this paper are constrained to be 0 in the X and Y positive and negative semi-axis directions and at the bottom, with the bottom being a static boundary, while the top is free in the Z-direction without constraint. The displacement boundary conditions are achieved by specifying the velocity component of each boundary as 0. The dynamic boundary condition is set after the ground stress equilibrium, and each stress value returns the initial stress state to 0.

The form of seismic wave used in this paper is a synthetic wave, acting at the bottom, applied along the tunnel's axial Y direction and perpendicular to the axial X direction, but discounted by 50% for the vertical tunnel axial X direction, according to the ratio of 0.5:1 with the seismic wave along the axial tunnel input. Considering the calculation time cost and accuracy control, seismic waves up to 5 s are selected in this paper. The dynamic analysis is solved by concentrating the real mass of the nodes in the grid density, and the critical calculation time steps during the calculation are as follows:

$$\Delta t_{critical} = \min \left\{ \frac{V}{C_p A_{max}^f} \right\} \quad (2)$$

where V is the volume of the tetrahedral subunit; C_p is the p-wave velocity; and A_{max}^f is the maximum surface area associated with the tetrahedral subcell.

The primary seismic intensity of the proposed project area is 7 degrees, the design primary seismic acceleration value is 0.10 g, the third group is selected in the seismic design grouping, and the site category is I_1 according to each geotechnical layer and overburden in the site area, and its characteristic period value is 0.35 s. This paper uses a form of local damping, and its value is taken as 0.2.

Underground rock formations are subject to fault motion after geological tectonic movements or extrusion by large ground stresses, producing significant displacements along the active fault zone. Due to the complexity of the stratigraphic dislocation, the dislocation is imposed after the tunnel excavation and support, and in this paper, three cases are considered for the direction factor.

- (1) Applying velocities for Y of 3×10^{-5} m/s and Z of 2×10^{-6} m/s to the Y = 0 surface.
- (2) Applying a velocity for Y towards 3×10^{-5} m/s to the Y = 0 surface.
- (3) A set of velocities for Y toward 3×10^{-5} m/s and 3×10^{-5} m/s are respectively applied to the two faces of the model, Y = 0 and Y = 120 m, in a squeezed manner.

3.4. Numerical Simulation Scheme

In this study, three major groups of eight working conditions for the simulation schemes are designed for the seismic dynamic and fault motion effects of the subsea tunnel crossing the active fault zone, and the tunnel lining stiffness is shown in Table 2.

Table 2. Simulation schemes considering different supporting structures, dynamic effects, and fault motions.

No.	Excavation Method	Factor I	Factor II	Factor III
1	Up and down step method	Lining II	/	/
2		Lining I	Seismic dynamic	/
3		Lining II		
4		Lining III		
5		Lining IV	Seismic dynamic	Orthogonal stagger (Y and Z directions)
6		Lining II		Unidirectional stagger (Y-direction)
7				Extrusion stagger (Y-direction)
8				

4. Results

Through the numerical simulation of multiple control variables, the stability analysis of the face of the subsea shield tunnel under different geological and construction conditions is carried out, and the influence of different factors on the face instability is studied.

4.1. Basic Features of Undersea Tunnel Excavation by up and down Step Method

The up-and-down step method excavates the upper and lower sections successively and advances simultaneously, which is conducive to the stability of the tunnel face. It has advantages over the full-section method in terms of simplicity of operation, low disturbance, and construction process safety. The displacement clouds after the upper and lower step method of excavation are shown in Figure 4, the acceleration clouds in Figure 5, and the stress clouds in Figure 6.

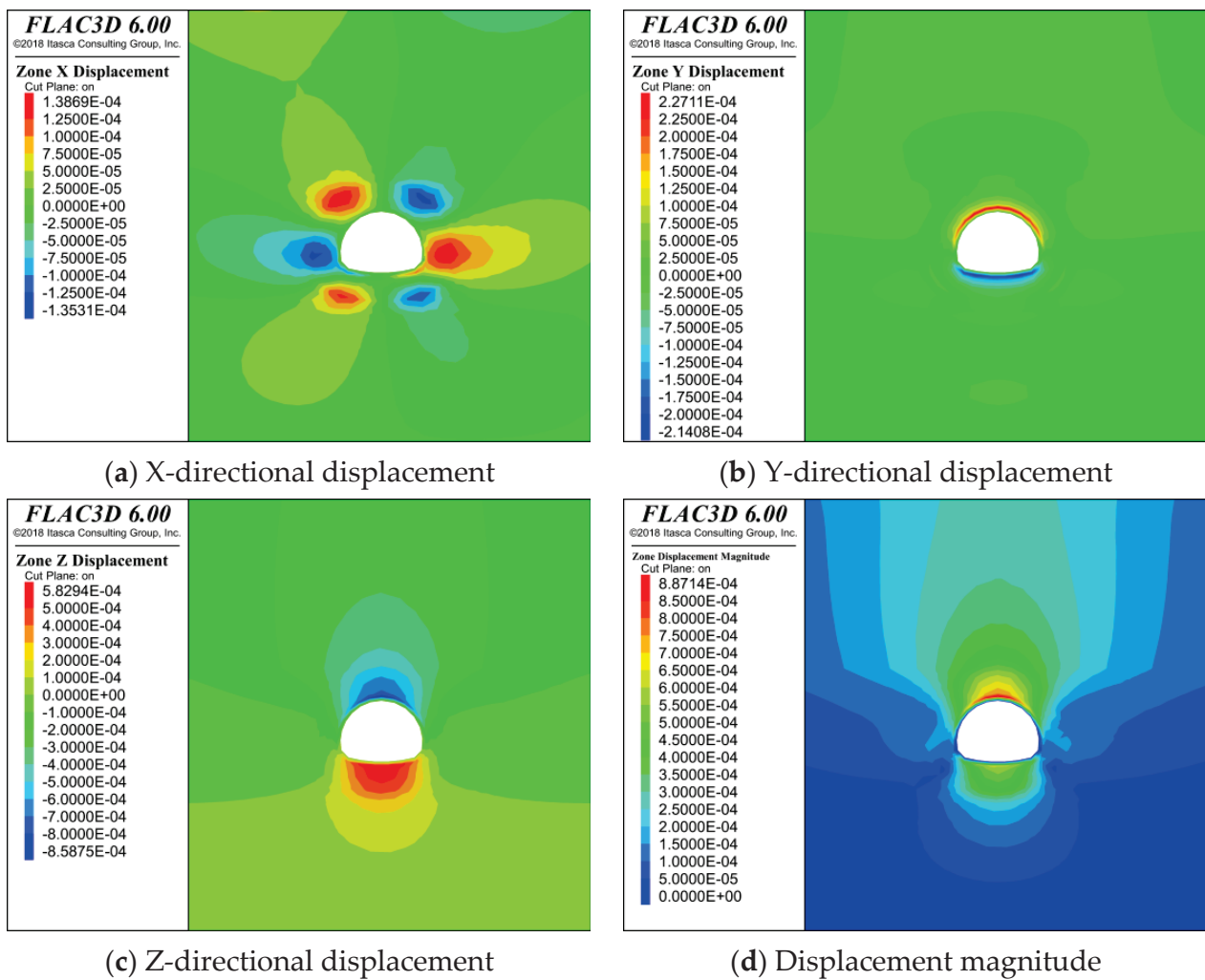


Figure 4. Displacement clouds of the tunnel section at $Y = 40$ m.

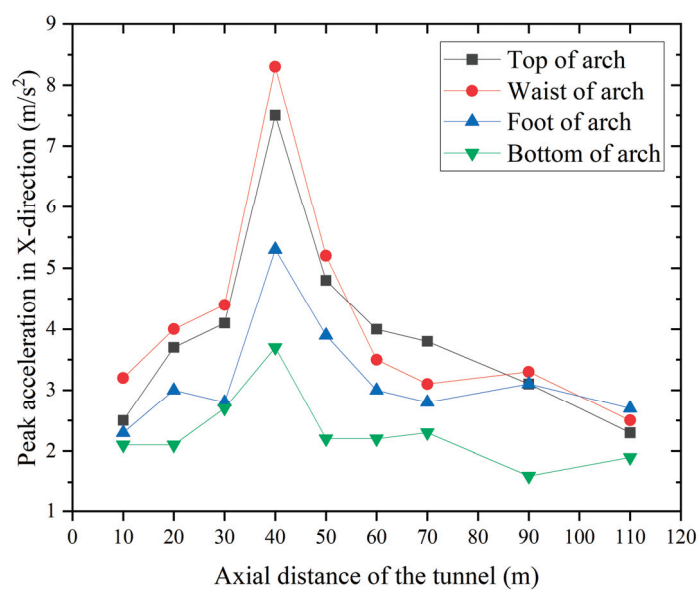


Figure 5. Distribution of peak acceleration in the X-direction along the axial direction of the tunnel.

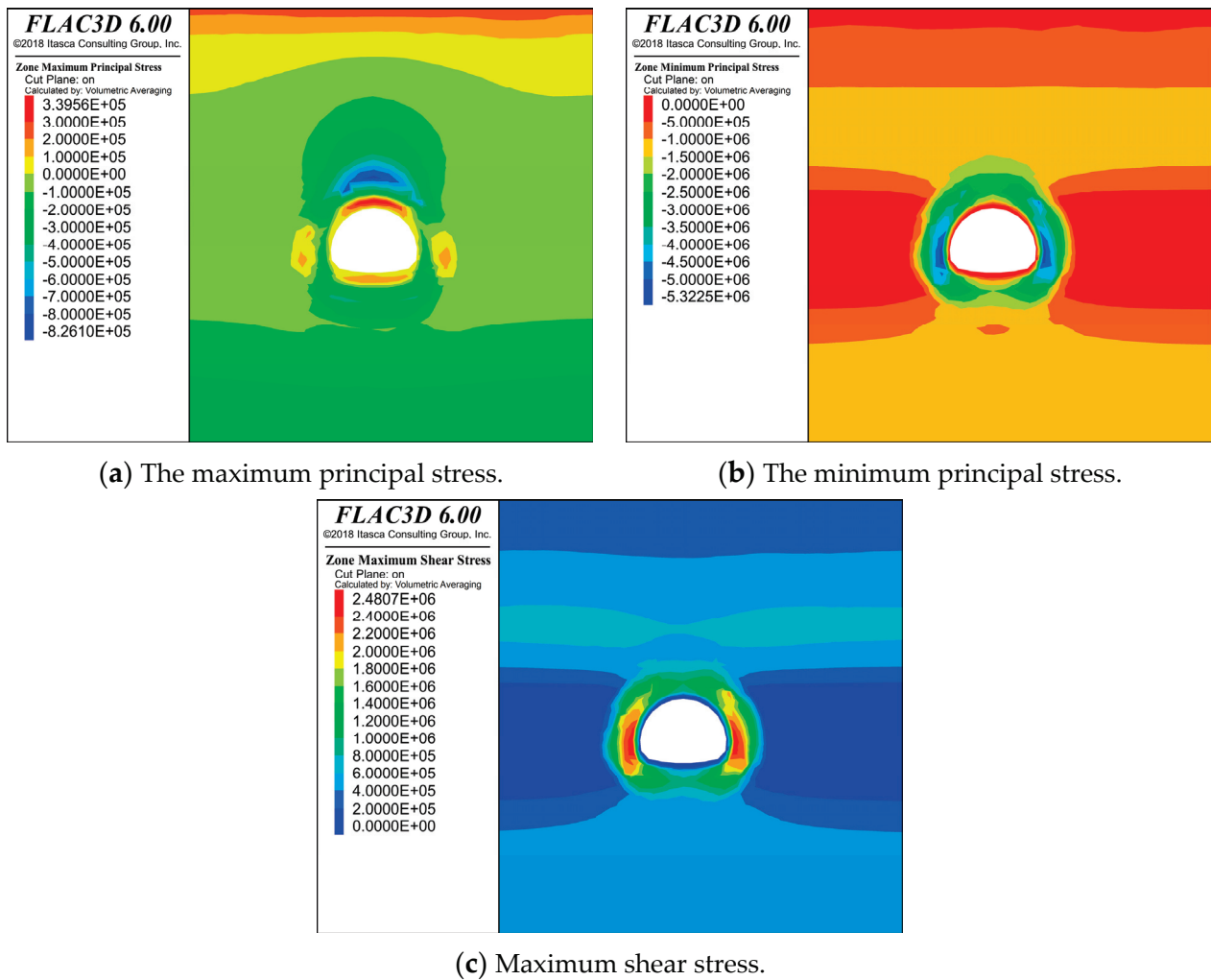


Figure 6. Stress clouds of the tunnel section at Y = 40 m.

Following excavation by the up-and-down step method, it can be obtained that the displacements generated by the tunnel are mainly settlements. The highest acceleration peaks are found at the top and waist of the tunnel. Based on the trend of the data from the measured points arranged in the tunnel section, it can be determined that the fault caused the greatest impact at Y = 40 m.

4.2. Dynamic Stability of Tunnel Structures with Different Support Parameters under Seismic Effects

Further, the dynamic stability is analyzed under different liner stiffnesses in comparison, and four groups of liner stiffnesses are set in increasing order, with the bulk moduli of 3.89 GPa, 7.78 GPa, 15.56 GPa, and 38.9 GPa; and shear moduli of 2.915 GPa, 5.83 GPa, 11.66 GPa, and 29.15 GPa, respectively.

In the total displacement cloud of the section at Y = 40 m (Figure 7), the total deformation trend is the same under different liner stiffnesses, and the total displacement decreases with the increase of liner stiffness. The results of each displacement component at Y = 40 m profile with different liner stiffnesses are shown in Table 3. The X-direction and Z-direction displacements decrease with increasing liner stiffness. Although the difference is not significant, the trend is consistent with the displacement magnitude. The peak stresses of different liner stiffnesses are demonstrated in Figure 8.

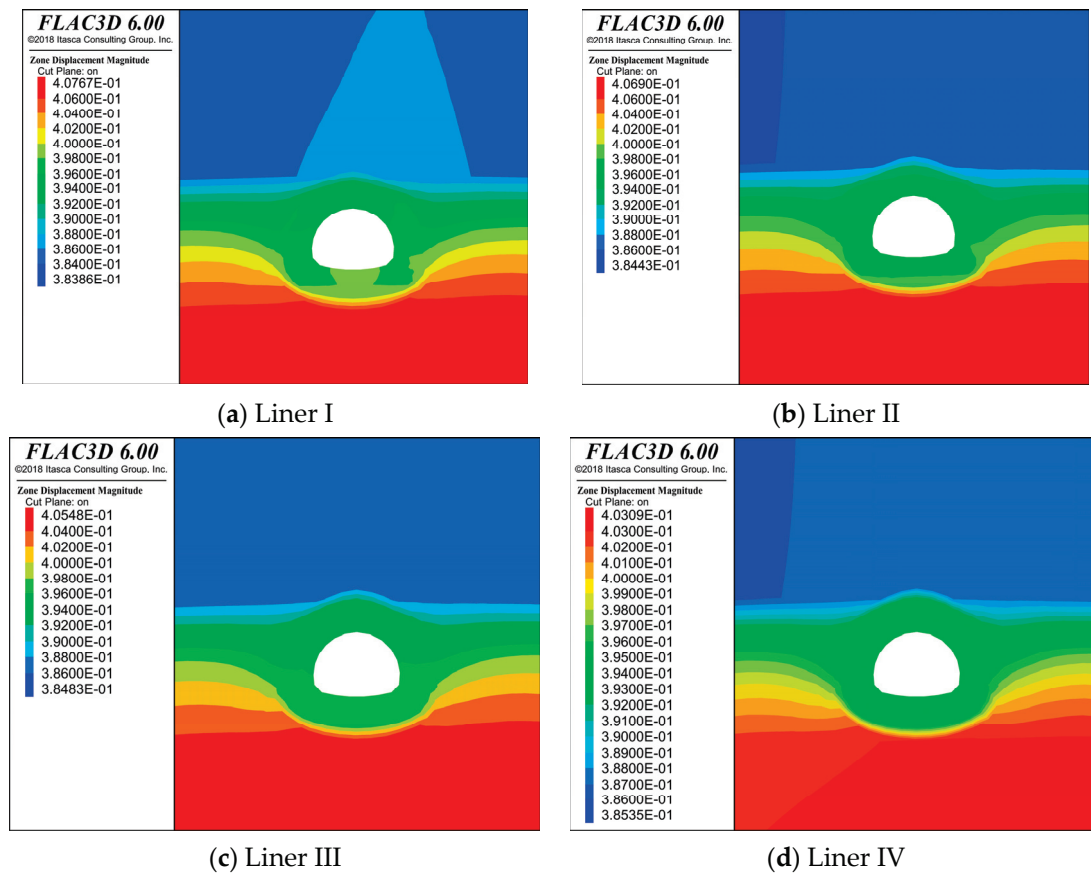


Figure 7. Displacement magnitude clouds of tunnel section at $Y = 40$ m with different lining stiffness.

Table 3. Maximum displacement of surrounding rock under different lining stiffness in the section at $Y = 40$ m.

Liner Type	X-Displacement/ cm	Y-Displacement/ cm	Z-Displacement/ cm	Displacement Magnitude/cm
Liner I	1.2035	2.0786	−40.701	40.767
Liner II	1.1865	2.0785	−40.624	40.690
Liner III	1.1608	2.0785	−40.481	40.548
Liner IV	1.1620	2.0785	−40.242	40.309

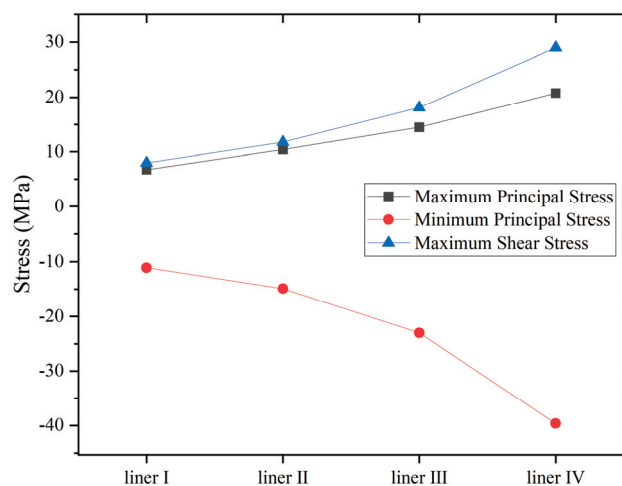


Figure 8. Peak stress of different lining stiffness.

In the case of specific initial support, the X- and Z-directional (reflected as settlement) displacements become smaller as the liner stiffness increases. The absolute values of maximum and minimum principal stress and maximum shear stress all increase with the increase of liner stiffness under particular initial support, indicating that its stress pattern changes and takes on more stress.

4.3. Dynamic Stability of Tunnel Structures under the Effects of Fault Motion

The forms of staggering in different ways are compared. From Figure 9, we can see that the peak displacement of the model lining structure is extrusion stagger > unidirectional stagger > orthogonal stagger, and the results show that the most unfavorable case for the lining structure should be the third case. The stress indicators (Figure 10) show that the maximum and minimum principal stresses and maximum shear stresses under orthogonal stagger and unidirectional stagger are tiny. In contrast, the stresses under phase stagger are much larger than the first two, reflecting the most unfavorable stresses on the lining structure. In the orthogonal and unidirectional staggering, compressive stresses appear in the parts of the maximum principal stress cloud where no velocity is applied, and tensile stresses appear at the ends of the tunnel in the minimum principal stress cloud. In the phase staggering, compressive stresses appear at the ends of the tunnel in the maximum principal stress cloud, and tensile stresses appear in the middle in the minimum principal stress cloud. This indicates that the fault motion greatly impacts the lining structure, and it also means that the lining in the subsea tunnel interval near the active fault zone has higher requirements.

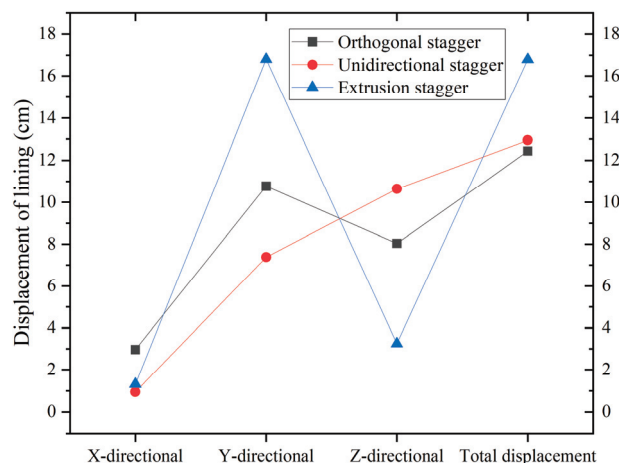


Figure 9. Lining displacement under three types of fault motion.

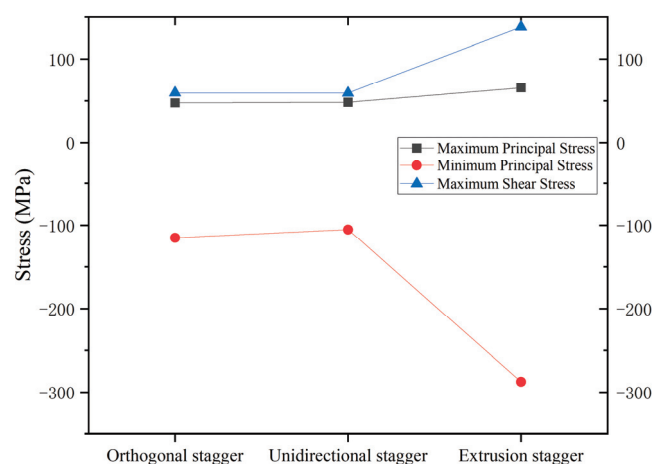


Figure 10. Peak stress under three types of fault motion.

Considering the factors of fault motion and seismic activity, since the direct cause of fault motion is the displacement and motion of the two plates, the direct-acting surface is the most critical part. The results also confirm that the maximum and minimum principal stresses and maximum shear stresses are on the direct-acting surface, and both sides are maximally under the effect of fault motion and bear most of the force, while the lining at the farther side bears less force. It should be that the active fault zone is the defense's focus.

5. Discussion and Limitations

When the active fault zone starts to misshape, the rock layers on both sides are subjected to shear or tensile compression, and relative displacement occurs between the upper and lower plates [9,14]. Furthermore, it contacts the seafloor overburden, transmitting shear stress to the far side. Lateral bending may occur, leading to damage on the seafloor surface. The activity of the active fault zone causes displacement and mechanical effects only in a specific range on both sides of the active fault zone, while it has less influence on the more distant rock masses. For different ways of movement and displacement of the fracture zone, its tension and compression states do not change. Mainly, the changes and differences are of specific values also affect the range and peak of its force-deformation area. When the pressure on both sides of the active fault zone is very high, the phenomenon of tension damage will also occur, further causing the separation and gap between the upper and lower plates of the active fault zone. The width and angle of the active fault zone do not change the tunnel structure's deformation, displacement, tension, and shear. In practice, this is often accompanied by earthquakes, cracking, depressions, or bulges in the ground that release energy from the deeper rocks.

Only when fault motion occurs are the tunnel and the surrounding rock subjected to tremendous stress values, and the peak acceleration and relative displacement at this time are also enormous. The cross-sea tunnel through the active fault zone is mainly affected by shear damage, which causes relative displacement and misalignment. This can cause displacement, deformation, and obvious dislocation of the tunnel structure, or even collapse and destruction of the whole, which poses a considerable risk of harm during construction and regular use. Therefore, special attention should be paid to exploring the specific location of the detailed active fault zone and its detailed parameters in the project's construction, and to strengthening the advance forecasts, real-time monitoring, and risk assessment to better cope with various risk hazards [19–21].

Liu et al. [2] showed the danger of the subsea tunnel face when crossing the Cangkou fault, and that the deformation of the tunnel face can be controlled by increasing the tunnel burial depth (but not exceeding the limit), reducing the tunnel diameter, and increasing the surrounding rock strength.

The active fault zone set in this study is tilted in a single direction after some simplification, while in practice, it is mainly tilted in two directions. In this study, the active fault zone is considered to suffer from the effects of seismic dynamics and fault motion under the separate working conditions. However, the more complex dual actions of simultaneous effects are not considered.

Based on this study, adding methods such as fluid–solid coupling and flow coupling to simulate the pore water pressure and rock force will bring the model closer to the actual situation. Designs including a flexible lining and damping layer can also be added to offset the large deformation caused by fault motion or dynamic seismic effects.

6. Conclusions

In this study, we use the second subsea tunnel in Jiaozhou Bay as the background and establish a finite difference model to analyze the force and deformation of the tunnel lining structure under the action of seismic dynamics and fault motion and obtain the following conclusions.

- (1) In the case of specific initial support, both X- and Z-directional (reflected as settlement) displacements become smaller with the increase of lining stiffness, but the

Y-directional displacement remains constant. The absolute values of the maximum and minimum principal stresses and maximum shear stress all increase with the increase of liner stiffness.

- (2) The active fault zone undergoes motion, and it can be known that larger displacements occur when specific external influences are applied. It is concluded that the most unfavorable situation will be caused when phase extrusion is applied. The multi-directional disturbance and external forces applied will make the rock formation more prone to motion, but the displacement values are not significant due to the presence of the lining structure.

Author Contributions: Z.L., Investigation, Writing—original draft, Funding acquisition. Q.L., Data curation, Formal analysis. S.L., Investigation. X.L., Validation. Y.Z. (Yongqiang Zhang), Visualization. S.J., Formal analysis. G.S., Writing—review and editing. Y.Z. (Yuting Zhao), Writing—review and editing, Supervision, Funding acquisition. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the Zhi-Qiang Li of Foundation of Key Laboratory of Geological Safety of Coastal Urban Underground Space (BHKF2021Z01), the Shandong Provincial Natural Science Foundation (ZR2022QD014), the Postdoctoral Innovation Project of Shandong Province (SDCX-ZG-202203030), and by Yuting Zhao of the Shandong Provincial Natural Science Foundation (ZR2022MD112, and ZR2020MD037), the Postdoctoral Innovation Project of Shandong Province (SDCX-ZG-202203030), the Laoshan Laboratory Science and Technology Innovation Project (LSKJ202203401 and LSKJ202203404), and the Qingdao Geo-Engineering Surveying Institute Project (2022-QDDZYKY04).

Data Availability Statement: Data is contained within the article.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Dammyr, Ø.; Nilsen, B.; Gollegger, J. Feasibility of tunnel boring through weakness zones in deep Norwegian subsea tunnels. *Tunn. Undergr. Space Technol.* **2017**, *69*, 133–146. [CrossRef]
2. Liu, Q.; Liu, Z.; Xue, Y.; Zhang, G.; Li, X.; Zhou, B.; Gong, H.; Li, Z. Deformation features and failure mechanism of subsea shield tunnels with different burial depths crossing fault-zone. *Mar. Georesources Geotechnol.* **2023**, 1–15. [CrossRef]
3. He, C.; Geng, P. Challenges and countermeasures of railway tunnel construction in macroseismic active Fault Zone. *China Railw.* **2020**, *12*, 61–68.
4. Wang, W.; Ren, Q. General introduction to the effect of active fault on deeply buried tunnel. *Earthq. Eng. Eng. Vib.* **2006**, *26*, 175–180.
5. Ma, S.; Zhang, L.; Wang, D.; Tan, X.; Li, S.; Liu, Y. Analysis of tunnel lining failure mechanism under the action of active fault. *Shock. Vib.* **2021**, *2021*, 9918021. [CrossRef]
6. Zhang, X.; Jiang, Y.; Maegawa, K. Mountain tunnel under earthquake force: A review of possible causes of damages and restoration methods. *J. Rock Mech. Geotech. Eng.* **2020**, *12*, 414–426. [CrossRef]
7. Zheng, Q.; Zhang, X.; Shen, Y.-S.; Qiu, J.-T.; Wang, Y.-D.; Chen, K.-F. Failure mechanism of segmental lining structure in fault-crossing tunnel: An experimental and numerical investigation. *J. Cent. South Univ.* **2023**, *30*, 2392–2410. [CrossRef]
8. He, C.; Li, L.; Zhang, J.; Geng, P.; Yan, Q.-x. Seismic damage mechanism of tunnels through fault zones. *Chin. J. Geotech. Eng.* **2014**, *36*, 427–434.
9. Fang, L.; Jiang, S.; Lin, Z.; Wang, F. Shaking table model test study of tunnel through fault. *Rock Soil Mech.* **2011**, *32*, 2709–2713.
10. Shahidi, A.; Vafaeian, M. Analysis of longitudinal profile of the tunnels in the active faulted zone and designing the flexible lining (for Koohrang-III tunnel). *Tunn. Undergr. Space Technol.* **2005**, *20*, 213–221. [CrossRef]
11. Yu, L.; Zhang, X.; Wang, M.; Yang, H.; Luo, X.; Qin, Y.; Tang, L. A simplified analytical method for the tunnels' mechanical behaviour under the action of active fault zones. *Comput. Geotech.* **2023**, *164*, 105789. [CrossRef]
12. Dalgıç, S. Tunneling in fault zones, Tuzla tunnel, Turkey. *Tunn. Undergr. Space Technol.* **2003**, *18*, 453–465. [CrossRef]
13. Kiani, M.; Ghalandarzadeh, A.; Akhlaghi, T.; Ahmadi, M. Experimental evaluation of vulnerability for urban segmental tunnels subjected to normal surface faulting. *Soil Dyn. Earthq. Eng.* **2016**, *89*, 28–37. [CrossRef]
14. Liu, X.; Li, X.; Sang, Y.; Lin, L. Experimental study on normal fault rupture propagation in loose strata and its impact on mountain tunnels. *Tunn. Undergr. Space Technol.* **2015**, *49*, 417–425. [CrossRef]
15. Corigliano, M.; Scandella, L.; Lai, C.G.; Paolucci, R. Seismic analysis of deep tunnels in near fault conditions: A case study in Southern Italy. *Bull. Earthq. Eng.* **2011**, *9*, 975–995. [CrossRef]
16. Roy, N.; Sarkar, R. A review of seismic damage of mountain tunnels and probable failure mechanisms. *Geotech. Geol. Eng.* **2017**, *35*, 1–28. [CrossRef]

17. Kamran, M.; Hu, X.; Awais Hussain, M.; He, K.; Nawaz, A.; Ali, R. Characterizing the Fundamental Controls on Deformation and Stability of an Active Reservoir Landslide, Southwest China. *Pol. J. Environ. Stud.* **2022**, *31*, 3611–3626. [CrossRef]
18. Qu, L.; Li, X.; Dai, Z. Jiaozhou bay subsea tunnels and underground tunnel project in Qianhai of Qingdao. *Tunn. Constr.* **2020**, *40*, 915–924.
19. Wang, S.; Li, L.-P.; Cheng, S.; Hu, H.; Jin, H.; Gao, S. Dynamic risk assessment method of tunnel collapse based on attribute interval assessment model and application. *Pol. J. Environ. Stud.* **2020**, *29*, 3853–3864. [CrossRef]
20. Li, S.; Liu, B.; Xu, X.; Nie, L.; Liu, Z.; Song, J.; Sun, H.; Chen, L.; Fan, K. An overview of ahead geological prospecting in tunneling. *Tunn. Undergr. Space Technol.* **2017**, *63*, 69–94. [CrossRef]
21. Nie, L.; Song, Z.; Zhou, W.; Li, Z.; Du, X.; Ma, L. Full-decay induced polarization multi-parameter tomography method for tunnel water inrush ahead prospecting: A case study. *Tunn. Undergr. Space Technol.* **2024**, *145*, 105589. [CrossRef]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Stress Concentration Factors Due to Misalignment at Girth Welds in Bi-Layer Pipes

Ruili Guo ¹, Hongyang Hu ², Haisheng Zhao ³ and Yao Zhang ^{2,*}

¹ School of Civil Engineering and Architecture, Wuhan Institute of Technology, Wuhan 430074, China; 20120103@wit.edu.cn

² School of Architecture and Civil Engineering, Xiamen University, Xiamen 361005, China; 25320221152407@stu.xmu.edu.cn

³ State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian 116024, China; hzhao@dlut.edu.cn

* Correspondence: zhangyao@xmu.edu.cn

Abstract: In recent years, bi-layer pipes, composed of an inner layer and an outer layer, have been widely used in offshore engineering. In this study, the governing equation for a bi-layer pipe subjected to axisymmetric loadings is derived based on classical shell theory. Then, the equation is used to develop stress concentration factor formulations for girth welds in bi-layer pipes with fabrication tolerances and thickness transitions. Axisymmetric finite element analysis is carried out to verify the accuracy of the proposed formulations. It is noted that these formulations can be well suited for determining the stress concentration factors for a wide range of thickness ratios (ratio of the inner layer thickness to the total thickness in a bi-layer pipe) varying from 0.0 to 1.0. They can also obtain accurate stress concentration factors whether the elastic modulus of the inner layer is smaller or larger than that of the outer layer.

Keywords: bi-layer pipe; finite element analysis; classical shell theory; stress concentration factor

1. Introduction

Misalignment widely exists in welded structures due to the eccentricities at fabrication tolerances or thickness transitions. The presence of misalignment introduces local bending stress at the welded joints, and in turn, the additional loading increases the stress of the structures. A stress concentration factor (SCF) [1] is used to assess quantitatively the effect of misalignment on the failures of structures, and it is defined as:

$$SCF = 1 + \frac{\sigma_s}{\sigma_{nom}} \quad (1)$$

where σ_s is the maximum induced bending stress due to misalignment and σ_{nom} is the nominal stress. The maximum stress, often referred to as the hot spot stress, is the sum of the maximum bending stress and the nominal stress. The hot spot stress, in combination with S-N data, is usually used for the fatigue life evaluation of structures subjected to dynamic loadings, and it is determined using the SCF multiplying the nominal stress. Therefore, it is necessary to calculate the SCFs for various plated and tubular structural members, and considerable research has been carried out to determine the SCFs [2–6]. Maddox [7] performed the derivation of SCFs at transverse butt welds under fatigue loadings. Based on extensive finite element (FE) analysis of various tubular configurations, a set of SCF formulations for the most commonly used tubular joints was developed by Efthymiou [8] and Smedley and Fischer [9]. Lotsberg [10–12] conducted a detailed assessment of SCFs for butt welds in plated and tubular structures subjected to axial force and internal pressure using the classical shell theory [13]. Effects of fabrication tolerances, thickness transitions and ring-stiffeners on SCFs were investigated systematically. It is

noted that these formulations are used to calculate the SCFs of homogeneous (one-layer) plated or tubular structures and have been incorporated into fatigue design rules such as DNV-RP-C203 [14] and BS 7910:2013 [15].

In recent years, bi-layer pipes, consisting of two layers with different functions for each layer, have been widely used to satisfy special needs [16,17]. For example, high-quality bi-layer pipelines are increasingly emerging as the most efficient mode of transporting vital energy resources such as oil and gas as extensive installation takes place [18–20]. The clad pipe is one of these cases, which is composed of a carbon steel outer pipe and a corrosion resistant alloy liner, protecting the outer pipe from corrosion. Despite the numerous numerical schemes [21–23] developed for assessing the fatigue and fracture behaviour of bi-layer pipes with misalignment at girth welds, their practical application remains challenging due to the difficulty of generating FE models in engineering settings. The SCF formulations, as mentioned earlier, encounter limitations when directly applied to evaluate stress concentrations in the notch regions at both the inner and outer sides of bi-layer pipes.

To overcome these challenges, this study adopts an innovative approach by initially deriving the governing equation for a bi-layer cylindrical shell under axisymmetric loadings, utilizing the principles of classical shell theory. This derived equation serves as the cornerstone for developing SCF formulations tailored explicitly for girth welds in bi-layer pipes, accounting for fabrication tolerances and thickness transitions. The efficacy and reliability of these formulations are rigorously validated through comprehensive axisymmetric FE analyses, ensuring their accuracy and applicability in real-world engineering scenarios. The developed models not only enhance the understanding of the fatigue and fracture behavior in bi-layer pipes but also provide a practical and robust tool for engineers dealing with complex structural assessments in the field.

2. Derivation of Governing Equation for a Bi-Layer Cylindrical Shell

A one-layer cylindrical shell subjected to the forces distributed symmetrically with respect to the axis of the cylinder is frequently encountered in practice. The governing equation for the shell has been built up using the classical shell theory [13]. In this study, a bi-layer cylindrical shell under the same loading conditions (Figure 1) is considered, and the corresponding governing equation is derived as an extension to the classical shell theory [13] (please refer to Appendix A for details). The definitions of related parameters used in Figure 1 are given in Table 1.

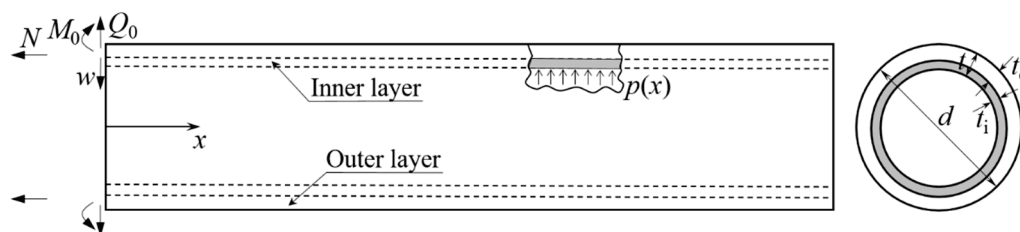


Figure 1. Bi-layer cylindrical shell subjected to symmetric loadings with respect to its axis.

Table 1. Definition of parameters.

Layers	Thickness	Outer Diameter	Radius	Elastic Modulus	Poisson's Ratio
Inner layer	t_i	$d - t_o$	/	E_i	ν_i
Outer layer	t_o	d	/	E_o	ν_o
Bi-layer pipe	$t = t_i + t_o$	d	$r = \frac{d-t}{2}$	/	/

The governing equation of the bi-layer shell presented in Appendix A is:

$$D_b \frac{d^4 w}{dx^4} + \frac{\alpha E_o t}{r^2} w = p(x) - \frac{v}{r} N \quad (2)$$

where D_b is the flexural rigidity of the bi-layer shell, α is a constant depending on the ratio of inner layer thickness to the total thickness and the ratio of elastic modulus of the inner layer to the elastic modulus of the outer layer, r is the radius of the bi-layer pipe, v is the Poisson's ratio for both inner layer and outer layer, and N is the axial force applied on the pipe. More details about D_b and α can be referred to in Appendix A.

By introducing the notation,

$$\beta^4 = \frac{\alpha E_o t}{4r^2 D_b} \quad (3)$$

Equation (2) can be expressed in the following simplified form:

$$\frac{d^4 w}{dx^4} + 4\beta^4 w = \frac{p(x)}{D_b} - \frac{v}{r D_b} N \quad (4)$$

The general solution of Equation (4), w , consists of a homogeneous part, w_h , and a particular part (w_p), and it is expressed as:

$$w = w_h + w_p \quad (5)$$

The homogeneous part of the solution is

$$w_h = \frac{e^{-\beta x}}{2\beta^3 D_b} \{ \beta M_0 [\cos(\beta x) - \sin(\beta x)] + Q_0 \cos(\beta x) \} \quad (6)$$

where M_0 and Q_0 are the bending moment and the shear force per unit circumferential length at $x = 0$, as shown in Figure 1.

Using the following notation,

$$\varphi(\beta x) = e^{-\beta x} [\cos(\beta x) + \sin(\beta x)] \quad (7a)$$

$$\psi(\beta x) = e^{-\beta x} [\cos(\beta x) - \sin(\beta x)] \quad (7b)$$

$$\theta(\beta x) = e^{-\beta x} \cos(\beta x) \quad (7c)$$

$$\zeta(\beta x) = e^{-\beta x} \sin(\beta x) \quad (7d)$$

Equation (6) can be written as:

$$w_h = \frac{1}{2\beta^3 D_b} [\beta M_0 \psi(\beta x) + Q_0 \theta(\beta x)] \quad (8)$$

The particular part of the general solution of Equation (5) is:

$$w_p = \frac{r}{\beta E_o t} [rp(x) - vN] \quad (9)$$

For constant p , the slope at a section x can be obtained by differentiating Equation (6), and this provides:

$$\frac{\partial w}{\partial x} = -\frac{1}{2\beta^2 D_b} [2\beta M_0 \theta(\beta x) + Q_0 \varphi(\beta x)] \quad (10)$$

The moment (per unit circumferential length) at a section x is determined by the following expression:

$$M_x = -D_b \frac{\partial^2 w}{\partial x^2} = \frac{1}{\beta} [\beta M_0 \varphi(\beta x) + Q_0 \zeta(\beta x)] \quad (11)$$

The shear force (per unit circumferential length) at a section x is calculated as:

$$Q_x = \frac{\partial M_x}{\partial x} = -[2\beta M_0 \zeta(\beta x) - Q_0 \psi(\beta x)] \quad (12)$$

3. Girth Welds in Bi-Layer Pipes with Fabrication Tolerances

In this section, a stress concentration in the girth weld of a bi-layer pipe with an eccentricity e as shown in Figure 2a is analyzed using the equation obtained in Section 2. Figure 2b shows the geometry analyzed. The length of the girth weld is L , and the corresponding slope is α . The pipe is subjected to an axial loading per unit circumferential length N . Due to the asymmetry of the static model, the moment at the middle of the weld ($x = 0$) is zero (inflection point), and the moments at the notch region ($x = \pm L/2$) have the same values ($Ne/2$) but different directions, as shown in Figure 2c.

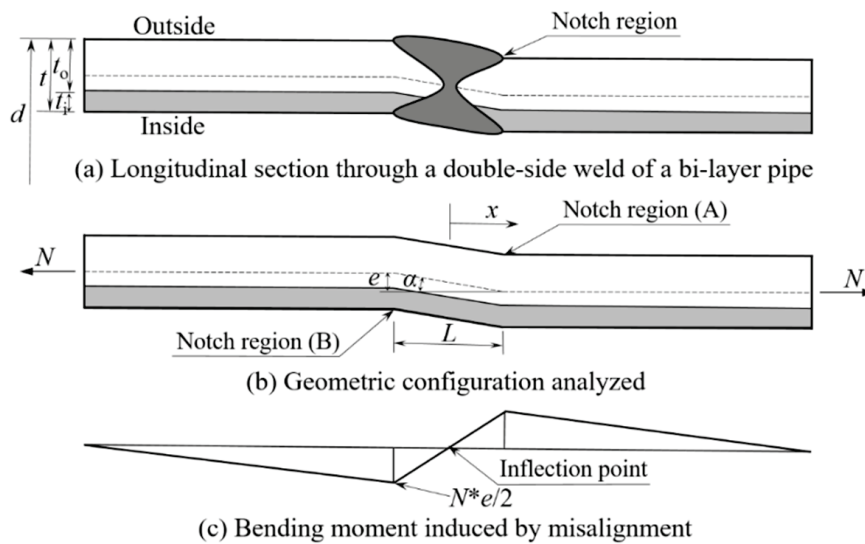


Figure 2. Model of a bi-layer pipe with fabrication tolerance over a section length L .

The boundary conditions of the bi-layer pipe shown in Figure 2b can be written as:

$$M|_{x=0} = 0, \quad Q|_{x=0} = -N \sin(\alpha) \cong -N \frac{e}{L} \quad (13)$$

On the other hand, according to Equation (11), the bending moment at $x = L/2$ is:

$$M|_{x=L/2} = -\frac{Ne}{\beta L} e^{-\frac{\beta L}{2}} \sin\left(\frac{\beta L}{2}\right) \quad (14)$$

In this study, the notch region considered is on the outside of the pipe indicated by “A” in Figure 2b. Hence, the bending stress induced by misalignment at “A” is:

$$\sigma_b^A = \frac{E_o N e (t - 2\eta_n)}{2(1 - \nu^2) D_b \beta L} e^{-\frac{\beta L}{2}} \sin\left(\frac{\beta L}{2}\right) \quad (15)$$

The nominal stress for the bi-layer pipe without misalignment needs to be determined in order to calculate the SCF. In Appendix A, it is assumed that the outer and the inner

layers of the aligned pipe undergo the same deformation when subjected to an axial loading N . Hence, the nominal stress is expressed as:

$$\sigma_{nom}^A = \frac{N}{(1 - \bar{t} + \bar{t}\bar{E})t} \quad (16)$$

where $\bar{t}$ is the ratio of inner layer thickness to the total thickness, and $\bar{E}$ is the ratio of the elastic modulus of the inner layer to the elastic modulus of the outer layer.

Based on Equations (15) and (16), the SCF at “A” is determined:

$$SCF = \frac{\sigma_{nom}^A + \sigma_b^A}{\sigma_{nom}^A} = 1 + \frac{3.22\mu}{(1 - \nu^2)^{1/4}} \frac{e}{L} \sqrt{\frac{d-t}{t}} e^{-\frac{\beta L}{2}} \sin\left(\frac{\beta L}{2}\right) \quad (17)$$

where

$$\mu = \frac{(1 - 2\bar{t} + \bar{t}^2 + 2\bar{t}\bar{E} - \bar{t}^2\bar{E})\sqrt{1 - \bar{t} + \bar{t}\bar{E}}}{(1 - 4\bar{t} + 4\bar{t}\bar{E} + 6\bar{t}^2 - 6\bar{t}^2\bar{E} - 4\bar{t}^3 + 4\bar{t}^3\bar{E} + \bar{t}^4 - 2\bar{t}^4\bar{E} + \bar{t}^4\bar{E}^2)^{3/4}} \quad (18)$$

When the Poisson’s ratio is set as 0.3 ($\nu = 0.3$), Equation (17) can be written as:

$$SCF = 1 + 3.30\mu \frac{e}{L} \sqrt{\frac{d-t}{t}} e^{-\frac{\beta L}{2}} \sin\left(\frac{\beta L}{2}\right) \quad (19)$$

The above expression is the same with the formulation derived by Lotsberg [10] for double-sided circumferential welds in tubular members except the remaining dimensionless parameter μ accounting for the varying thickness ratio $\bar{t}$ and elastic property ratio $\bar{E}$. When $\bar{t} = 0$ or $\bar{E} = 1$, $\mu = 1$, and Equation (19) for calculating the SCF of a misaligned bi-layer pipe is degraded to the formulation for a one-layer pipe given by Lotsberg [10], verifying the validity of Equation (19) derived in this study to some extent.

The comparison of the SCFs obtained by FE analysis and Equation (19) is carried out to further demonstrate the accuracy of the formulation. The mesh design of the FE models is based on the recommendations for the notch stress methodology in Appendix D of DNV-RP-C203 [14]. The notches at “A” and “B” are modeled using a radius of 1.0 mm. An axisymmetric pipeline model with a diameter of 400 mm and thickness of 20 mm is analyzed in this study.

At first, the SCFs obtained by FE analysis for a misaligned one-layer pipe are compared with that given by Lotsberg [10] formulation to test the validity of FE models (Figure 3). It is observed that the maximum percentage difference of SCFs is 1.2% ($e/t = 0.15$), illustrating a good agreement between the FE results and the formulation. Hence, the FE models are considered suitable for numerical analysis of a misaligned bi-layer pipe when adequate elements are used to obtain good convergence. The convergent test shows that a total of nearly 1×10^4 elements is adequate in this study.

Figure 4 compares the SCFs from Equation (19) and FE analysis for varying e/t ($0.00 \leq e/t \leq 0.15$) and $\bar{E}$ ($\bar{E} = 0.5$ and 2.0) with constant $\bar{t}$ ($\bar{t} = 0.5$), showing a very good correlation between the two results. As the misalignment increases, the value of SCF also increases. FE analyses and formula-based calculations show consistency in the overall trend. However, it is observable that compared to FE results, the outcomes derived from formulas are slightly conservative. The reason might be that it is assumed that the shell has infinite curvature radius or the ratio of thickness to radius can be neglected in the theoretical derivation; however, the curvature radius is not infinite, and the ratio of thickness to radius is not zero although it is quite near zero in the FE models. Overall, both methods exhibit a very good correlation. This alignment in trends between the two approaches underscores the reliability of the derived formula in predicting the impact of misalignment on SCF in engineering applications.

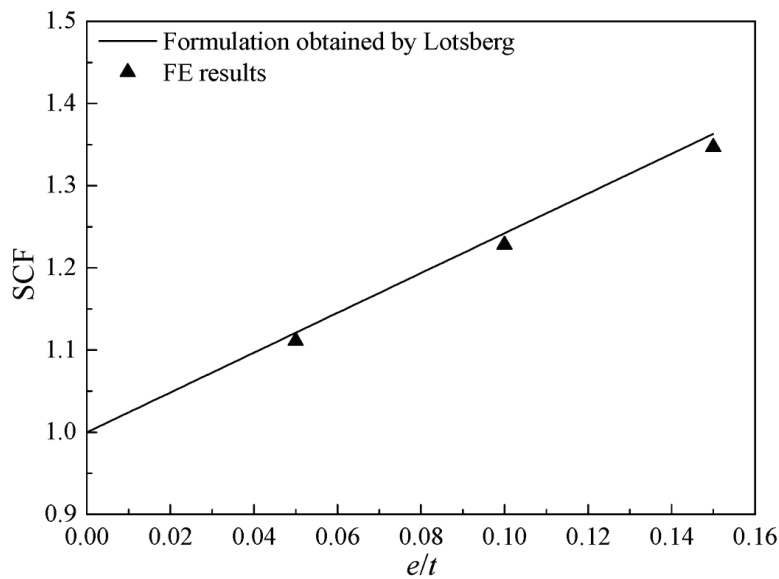


Figure 3. Comparison of SCFs from FE analysis and Lotsberg's formulation.

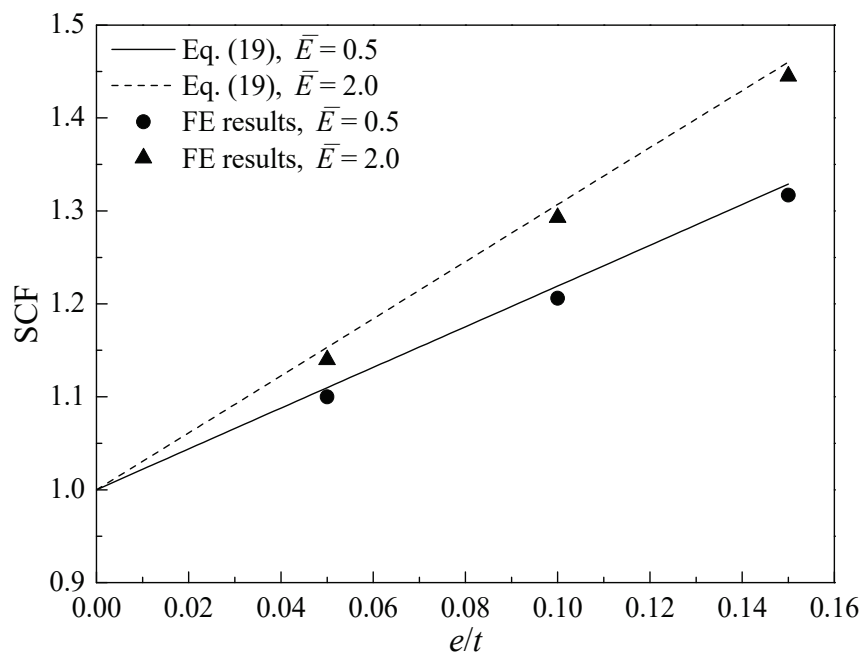


Figure 4. Comparison of SCFs from Equation (19) and FE analysis for $\bar{t} = 0.5$.

For varying $\bar{t}$ with constant e/t ($= 0.15$) and $\bar{E}$ ($= 0.5$), a good agreement between Equation (19) and FE results is also observed, as indicated in Figure 5. Therefore, Equation (19) can accurately assess the stress concentration in the notch region of the girth weld in a bi-layer pipe.

Figure 6 shows the influence of $\bar{E}$ with the variation of $\bar{t}$ on the SCF given by Equation (19) for constant e/t ($= 0.15$). $\bar{E} = 1$ indicates that the inner layer and the outer layer in the bi-layer pipe have the same elastic modulus, which is the case of the one-layer pipe. Hence, the SCF is not affected by the varying $\bar{t}$ as presented by a horizontal line in Figure 6. $\bar{E} > 1$ means the elastic modulus of the inner layer is higher than that of the outer layer, and a larger $\bar{E}$ raises a more pronounced effect on the SCF. For the case of $\bar{E} < 1$, a reverse trend is observed in comparison with that of $\bar{E} > 1$.

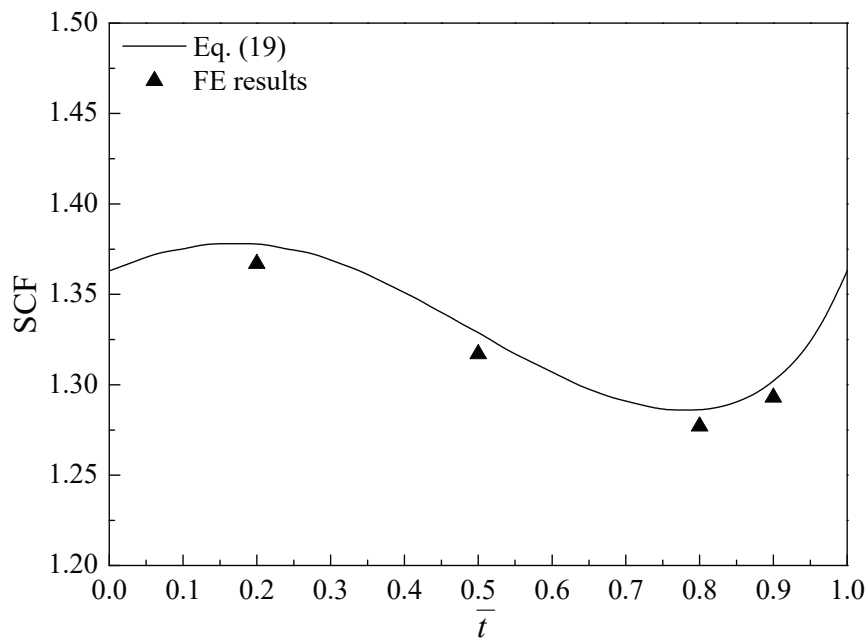


Figure 5. Comparison of variation of SCFs over $\bar{t}$ from Equation (19) and FE analysis for $e/t = 0.15$ and $\bar{E} = 0.5$.

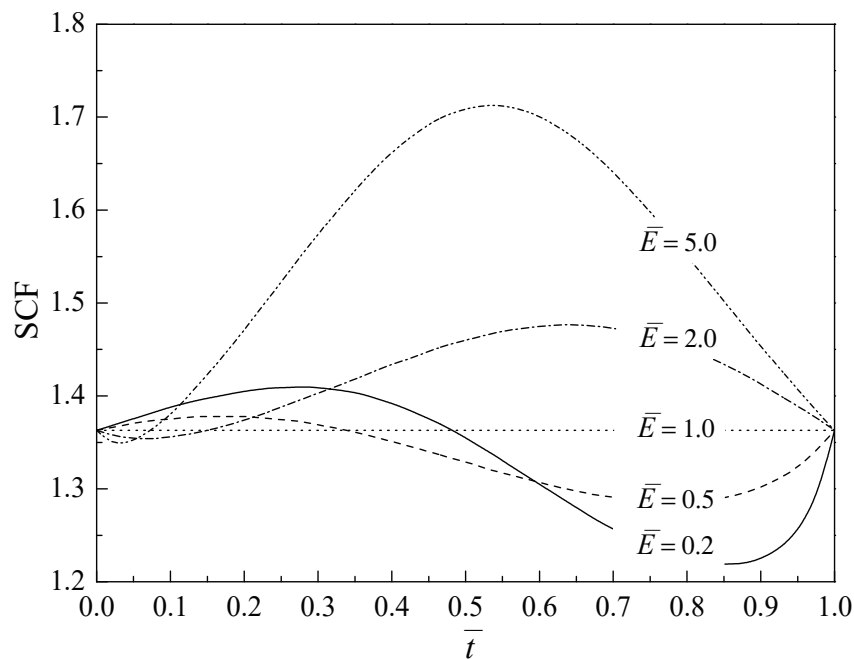


Figure 6. Effect of $\bar{E}$ on SCF over $\bar{t}$ for $e/t = 0.15$.

Even though it has been indicated that Equation (19) is used to calculate the SCF at the notch region “A”, it can be transformed into determining the SCF at “B” (Figure 2b) by substituting the two dimensionless parameters $(1 - \bar{t})$ and $1/\bar{E}$ into Equation (19) instead of $\bar{t}$ and $\bar{E}$. Figure 7 shows the calculation of the SCF at “B” for $\bar{E} = 2.0$ and $e/t = 0.15$ based on the above approach. The two parameters $(1 - \bar{t})$ and $1/\bar{E}$ ($= 0.5$) are substituted into Equation (19) and the plot of SCF over $\bar{t}$ for “B” is given using a dashed line in Figure 7. It is noted that the two lines for “A” and “B” are symmetric with $\bar{t} = 0.5$, and the validity of the above transform approach is further verified using the FE analysis as indicated by the triangle symbol in Figure 7.

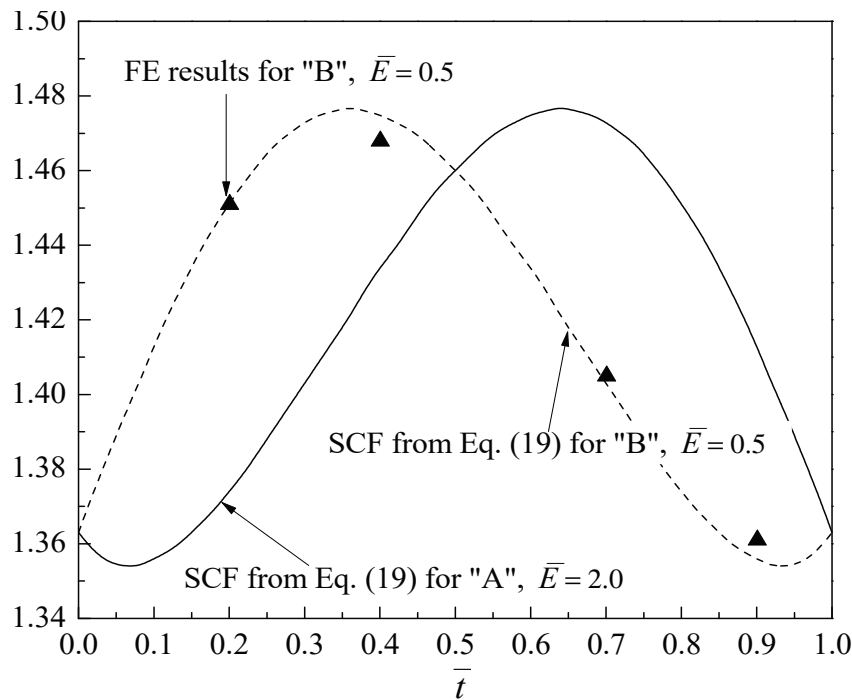


Figure 7. Calculation of SCF at notch region “B” based on that of “A” for $e/t = 0.15$.

The SCF formulation in Equation (19) is derived based on the double-side weld configuration of the bi-layer pipe with centerline offset misalignment. However, the formulation can also be applied to determine the SCF at the pipe’s notch region “A” with single-side welds (Figure 8). The SCF at “B” becomes close to 1.0 due to the region close to the intersection point where the bending moment induced by misalignment is zero.

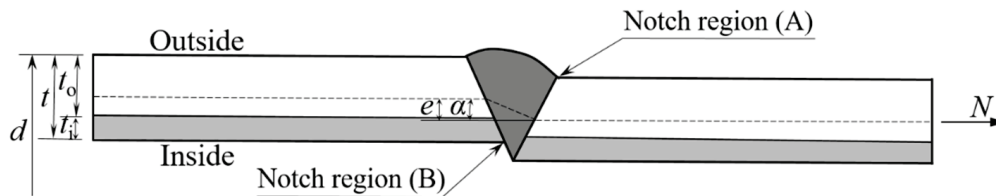


Figure 8. Longitudinal section through single-side welds of a bi-layer pipe.

4. Girth Welds in Bi-Layer Pipes at Thickness Transitions

This section details the derivation of the SCF in the girth weld of a bi-layer pipe with thickness transitions, as shown in Figure 9.

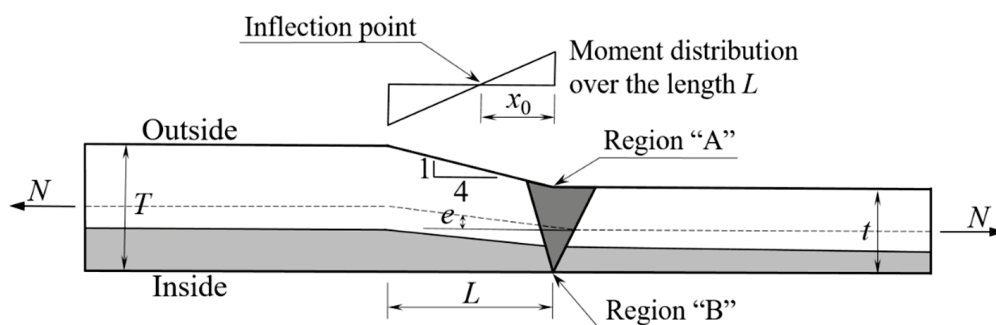


Figure 9. Longitudinal section of a bi-layer pipe with thickness transitions.

For the pipes subjected to dynamic loading, thickness transitions are normally to be fabricated with slope 1:4. The inflection point (zero moment) in Figure 9 needs to be determined in advance in order to calculate the eccentricity moment at the notch region of the girth weld based on Equation (11). It is assumed that the thickness ratio $\bar{t}$ keeps constant along the longitudinal section of the pipe. Thus, the rotational stiffness is in proportion to the pipe thickness t raised in a power of 2.5 according to Equation (10) and x_0 (please refer to Figure 9) is given as:

$$x_0 = L \frac{t^{2.5}}{t^{2.5} + T^{2.5}} \quad (20)$$

Following the same steps for calculating the SCF in Section 3, the SCF at the transition at the outside of the pipe (region “A” as indicated in Figure 9) is:

$$\text{SCF} = 1 + 6\mu_o \frac{e}{t} \frac{1}{1 + (T/t)^{2.5}} e^{-\beta x_0} \quad (21)$$

where

$$\mu_o = \frac{(1 - 2\bar{t} + \bar{t}^2 + 2\bar{t}\bar{E} - \bar{t}^2\bar{E})(1 - \bar{t} + \bar{t}\bar{E})}{(1 - 4\bar{t} + 4\bar{t}\bar{E} + 6\bar{t}^2 - 6\bar{t}^2\bar{E} - 4\bar{t}^3 + 4\bar{t}^3\bar{E} + \bar{t}^4 - 2\bar{t}^4\bar{E} + \bar{t}^4\bar{E}^2)} \quad (22)$$

The SCF at the transition at the interior side of the pipe (region “B” as indicated in Figure 9) is:

$$\text{SCF} = 1 - 6\mu_i \frac{e}{t} \frac{1}{1 + (T/t)^{2.5}} e^{-\beta x_0} \quad (23)$$

where

$$\mu_i = \frac{(1 - \bar{t}^2 + \bar{t}^2\bar{E})(1 - \bar{t} + \bar{t}\bar{E})}{(1 - 4\bar{t} + 4\bar{t}\bar{E} + 6\bar{t}^2 - 6\bar{t}^2\bar{E} - 4\bar{t}^3 + 4\bar{t}^3\bar{E} + \bar{t}^4 - 2\bar{t}^4\bar{E} + \bar{t}^4\bar{E}^2)} \quad (24)$$

When $\bar{t} = 0$ or $\bar{E} = 1$, μ_o and μ_i are both equal to the unit, and Equations (21) and (23) for calculating the SCF of a misaligned bi-layer pipe become the formulations for a one-layer pipe presented by Lotsberg [10], verifying the accuracy of the proposed formulations. In addition, FE analysis is carried out to further test the validity of the formulations, as done in Section 3. Figure 10 compares the SCFs from the present formulations and FE analysis. It is observed that the formulations are in good agreement with the numerical results except for the case of the SCF at region “B” for $\bar{E} = 0.5$. However, the present formulations show a trend closer to the FE results in comparison with that given by Lotsberg [10], as indicated by an arrow line in Figure 10. In general, the results obtained from the present formulations and FE analysis are consistent with one another, validating the accuracy of Equations (21) and (23) derived using the bi-layer cylindrical shell equation given in Section 2.

Figure 11 shows the effect of $\bar{E}$ on the SCFs with the increasing $\bar{t}$ for constant $T/t (= 2.0)$ at region “A” and “B”. The variation trend of the SCF with the varying $\bar{E}$ presented in Figure 11a is the same as that of Figure 9 for region “A”. However, a reverse trend is observed for region “B” (Figure 11b) compared with region “A” (Figure 11a), which can be easily explained by the opposite signs in Equations (21) and (23). The SCF at region “B” is minimal when region “A” has the maximum SCF value for certain dimensionless parameters $\bar{t}$ and $\bar{E}$, illustrating a reverse variation trend as shown in Figure 11a,b.

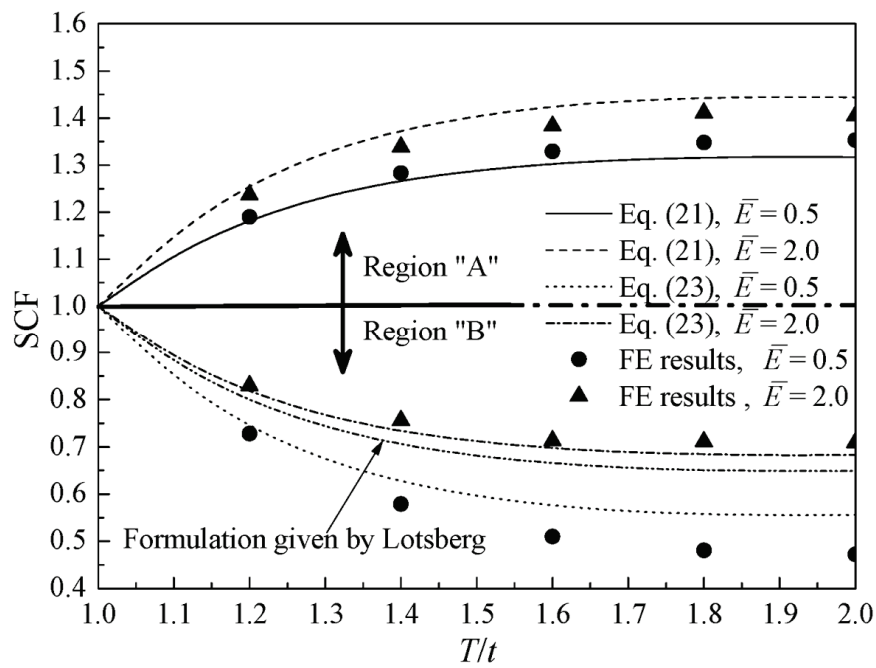


Figure 10. Comparison of SCFs from present formulations and FE analysis for $\bar{t} = 0.5$.

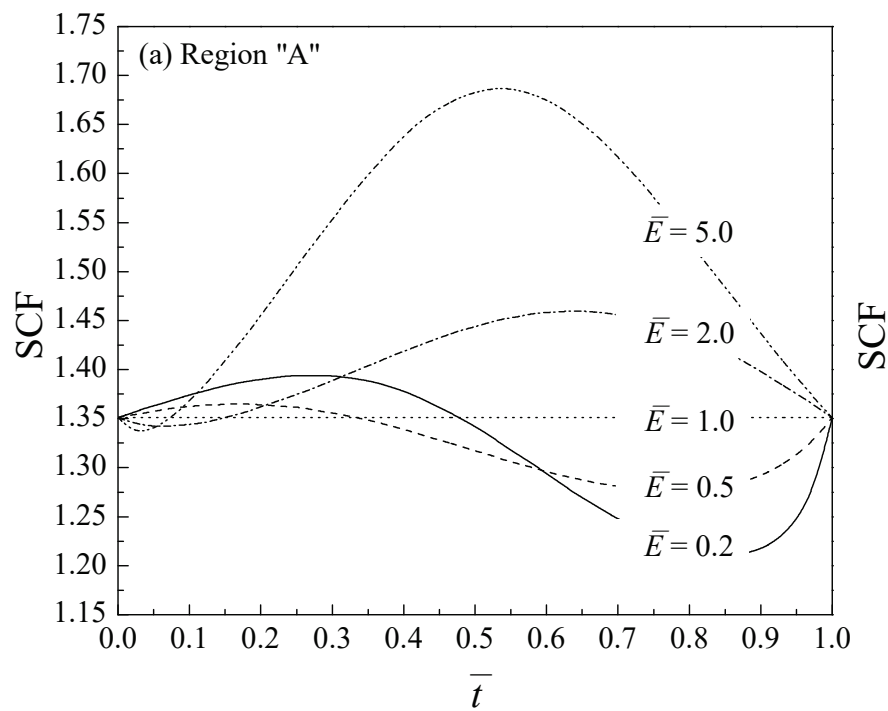


Figure 11. Cont.

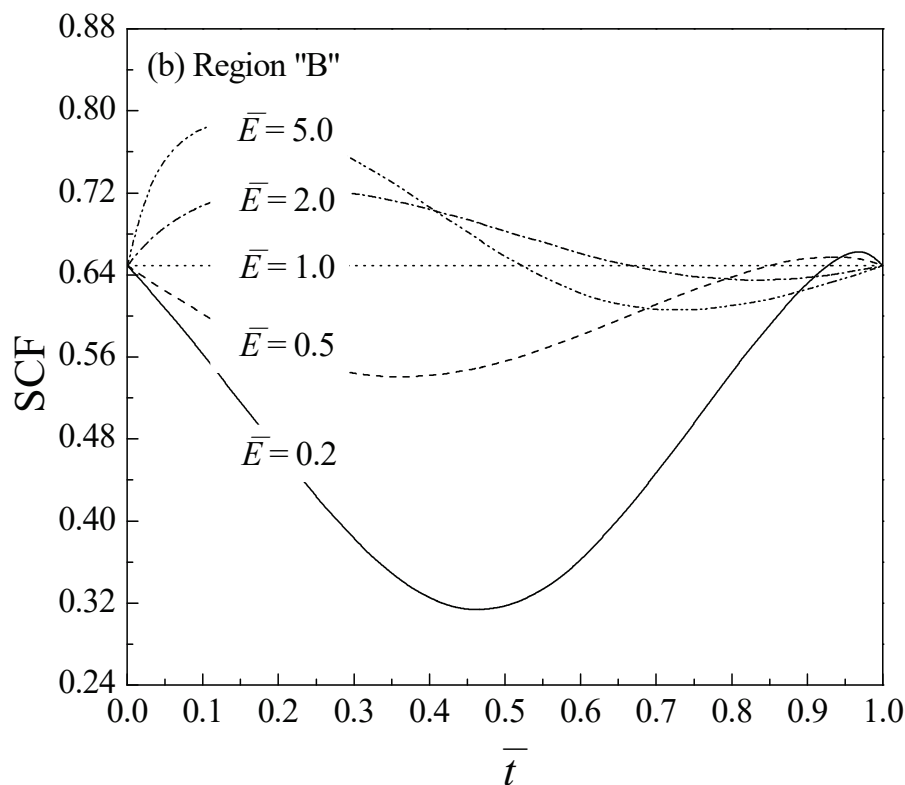


Figure 11. Effect of $\bar{E}$ on SCFs over $\bar{t}$ for $T/t = 2.0$.

5. Conclusions

The present study investigates the effect of fabrication tolerances and thickness transitions on the SCFs for girth welds in a bi-layer pipe. Based on the classical shell theory [14], the governing equation of a bi-layer cylindrical shell subjected to an axial force and an internal pressure is built up. Then, the equation is used to derive SCF formulations for girth welds of a bi-layer pipe with fabrication tolerances and thickness transitions. Axisymmetric FE analysis is carried out to validate the proposed formulations, and a good agreement between the formulation and the FE results is observed. These formulations are well suited for a wide range of thickness ratios $\bar{t}$ and elastic modulus ratios $\bar{E}$, and degraded to the expressions derived by Lotsberg [10] when $\bar{t} = 0$ or $\bar{E} = 1$. Finally, the influence of varying $\bar{t}$ and $\bar{E}$ on the SCF are investigated in detail.

Author Contributions: Conceptualization, Y.Z. and H.Z.; methodology, H.Z.; software, H.H.; validation, R.G. and H.H.; formal analysis, R.G. and H.H.; investigation, R.G. and H.H.; resources, Y.Z. and H.Z.; data curation, R.G. and H.H.; writing—original draft preparation, H.H. and H.Z.; writing—review and editing, R.G. and Y.Z.; visualization, H.Z.; supervision, Y.Z. and H.Z.; project administration, R.G.; funding acquisition, R.G. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the Scientific Research Foundation of the Wuhan Institute of Technology, grant number 21QD37.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

Appendix A

This appendix details the derivation of the governing equation for the bi-layer cylindrical shell subjected to an internal pressure $p(x)$ and an axial force N uniformly distributed along the x -direction, as shown in Figure 1. The total thickness of the shell is denoted by t , this quantity always being considered small compared to other dimensions of the shell. The neutral surface is the surface free of stress when a shell is loaded by a bending force, and the surface that bisects the thickness of the shell is named the middle surface. For the one-layer shell, it is noted clearly that the neutral surface is overlapped with the middle surface. However, the neutral surface of the bi-layer shell deviates from the middle surface due to the variation of material properties along the thickness direction. Therefore, it is necessary to determine the position of the neutral surface in order to derive the final governing equation.

Due to t being very small in comparison with the radius of the shell as referred to above, the infinitely small element (Figure A1a) cut from the shell can be regarded as a rectangular one (Figure A1b), and the small rectangular element follows the Kirchhoff assumptions [13] during the action of symmetric loadings. Therefore, the strain along the x -direction is distributed linearly, as indicated in Figure A1c, along the thickness direction, and it can be expressed as:

$$\varepsilon_x = \frac{\varepsilon_t}{t/2 - \eta_n} \eta \quad (\text{A1})$$

where ε_t is the x -direction strain at the top of the element, η is the vertical coordinate in the local ξ - η coordinate system, η_n denotes the vertical coordinate of the neutral surface. It is noteworthy that if the ratio of thickness to radius cannot be neglected, the above assumption may not be valid.

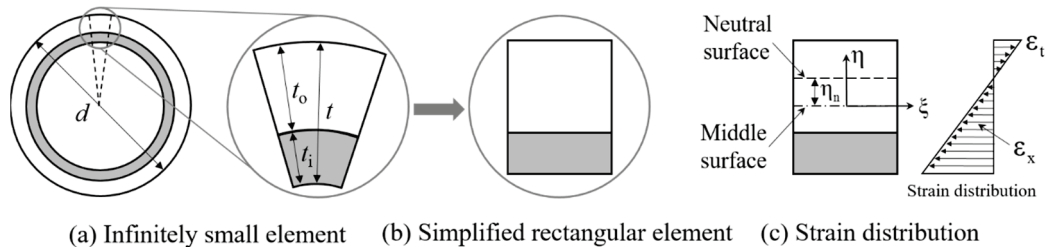


Figure A1. Schematics of small element cut from shell cross-section.

In this study, it is assumed that the shell's outer and inner layers have the same Poisson's ratio ($\nu_o = \nu_i = \nu$) but different elastic modulus ($E_o \neq E_i$). The position of the neutral surface can be determined based on the x -direction equilibrium of forces, and the integral expression is written as:

$$\int_0^{t/2-\eta_n} E_o \varepsilon_x d\eta = \int_{-(t/2+\eta_n)+t_i}^{-(t/2+\eta_n)+t_i} E_i \varepsilon_x d\eta + \int_{-(t/2+\eta_n)+t_i}^0 E_o \varepsilon_x d\eta \quad (\text{A2})$$

By solving the above integral, η_n is expressed in the following form:

$$\eta_n = \bar{\eta}_n t \quad (\text{A3})$$

where

$$\bar{\eta}_n = \frac{\bar{t}(1-\bar{t})(1-\bar{E})}{2(1-\bar{t}+\bar{t}\bar{E})}, \quad \bar{t} = t_i/t, \quad \bar{E} = E_i/E_o \quad (\text{A4})$$

and $\bar{t}$ indicates the ratio of the inner layer thickness to the total thickness, and $\bar{E}$ is the ratio of the inner layer elastic modulus to that of the outer layer, both characterizing the bi-layer shell, which is different from the one-layer one.

In the following, the governing equation of the bi-layer shell is built up under an axial force and an internal pressure. An infinitely small element (Figure A2) cut from the shell is analyzed to establish the equations of equilibrium required for this problem. N_x and N_φ are the axial forces per unit circumferential length and per unit length around the circumference of the shell, respectively, Q_x is the shear force per unit circumferential length, M_x and M_φ are the bending moments per unit circumferential length and per unit length around the circumference of the shell, respectively. Due to symmetry, other forces such as $N_{x\varphi}$, Q_φ and $M_{x\varphi}$ vanish in this case. Assuming that an internal pressure $p(x)$ normal to the surface and an axial force N uniformly distributed along the x -direction are applied, the equations of equilibrium obtained are:

$$\begin{aligned} N_x &= N \\ \frac{dQ_x}{dx} + \frac{1}{r} N_\varphi &= -p(x) \\ \frac{dM_x}{dx} - Q_x &= 0 \end{aligned} \quad (A5)$$

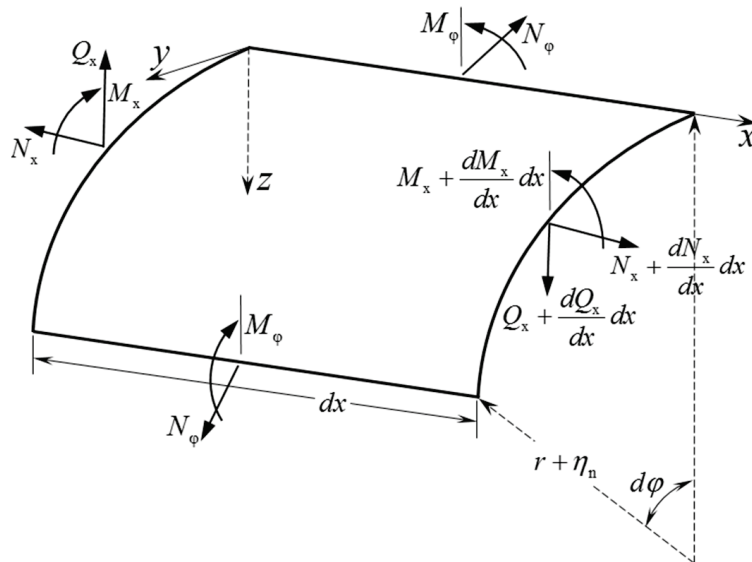


Figure A2. An infinitely small shell element used in the analysis.

The displacements of the points in the neutral surface of the shell need to be considered to solve the other three unknown quantities: N_φ , Q_x , and M_x . From symmetry, the displacement component v in the circumferential direction vanishes. The components u and w in the x - and z - directions are only considered. Therefore, the geometric equations can be written in the following simplified form:

$$\varepsilon_x = \frac{du}{dx}, \quad \varepsilon_\varphi = -\frac{w}{r} \quad (A6)$$

By using Hooke's law,

$$\sigma_x = \frac{E}{1-\nu^2} (\varepsilon_x + \nu \varepsilon_\varphi), \quad \sigma_\varphi = \frac{E}{1-\nu^2} (\varepsilon_\varphi + \nu \varepsilon_x) \quad (A7)$$

and the relationship between N_x , N_φ , Q_x , M_x and σ_x , σ_φ ,

$$N_x = \int_0^t \sigma_x d\eta, \quad N_\varphi = \int_0^t \sigma_\varphi d\eta, \quad M_x = \int_{-t/2-\eta_n}^{t/2-\eta_n} \sigma_x \eta d\eta \quad (A8)$$

the constitutive equations for this problem are:

$$\begin{aligned} N_x &= \alpha_1 \frac{du}{dx} - \alpha_2 \frac{w}{r} \\ N_\varphi &= -\alpha_1 \frac{w}{r} + \alpha_2 \frac{du}{dx} \\ M_x &= -D_b \frac{d^2 w}{dx^2} \end{aligned} \quad (\text{A9})$$

where

$$\alpha_1 = \frac{(1 - \bar{t} + \bar{t}\bar{E})tE_o}{1 - v^2}, \quad \alpha_2 = \frac{(1 - \bar{t} + \bar{t}\bar{E})tE_ov}{1 - v^2} \quad (\text{A10})$$

and D_b is the flexural rigidity of the bi-layer shell, which is expressed as:

$$D_b = \frac{E_o t^3}{24(1 - v^2)} \left[(1 - 2\bar{\eta}_n)^3 + \bar{E}(1 + 2\bar{\eta}_n)^3 + (1 + 2\bar{\eta}_n - 2\bar{t})^3 - \bar{E}(1 + 2\bar{\eta}_n - 2\bar{t})^3 \right] \quad (\text{A11})$$

Substituting Equation (A9) into Equation (A5) obtains the required governing equations,

$$\frac{d^2}{dx^2} \left(D_b \frac{d^2 w}{dx^2} \right) + \frac{\alpha E_o t}{r^2} w = p(x) - \frac{v}{r} N \quad (\text{A12})$$

where

$$\alpha = 1 - \bar{t} + \bar{t}\bar{E} \quad (\text{A13})$$

When the thickness of the shell is constant, the above equation is simplified as:

$$D_b \frac{d^4 w}{dx^4} + \frac{\alpha E_o t}{r^2} w = p(x) - \frac{v}{r} N \quad (\text{A14})$$

Obviously, the form of the equation is nearly the same as that of the one-layer shell except for the expressions of the flexural rigidity D_b and the dimensionless parameter α . When $\bar{t} = 0$ or $\bar{E} = 1$, $D_b = Et^3/12(1 - v^2)$ and $\alpha = 1$, and Equation (A14) becomes the governing equation for the one-layer shell derived based on the classical shell theory.

Appendix B

This appendix focuses on the generation of FE models for bi-layer pipelines with misalignment at girth welds, including a detailed description on the number of elements, types of elements, element sizes, loads, boundary conditions, and the overall structure of the finite element mesh. Moreover, it outlines the procedural steps and critical considerations in the generation of FE models.

Firstly, the FE model of bi-layer pipe without misalignment should be generated, which is actually not difficult in practice because the shape of bi-layer pipe is quite regular. Figure A3 shows the typical FE meshes of the girth weld.

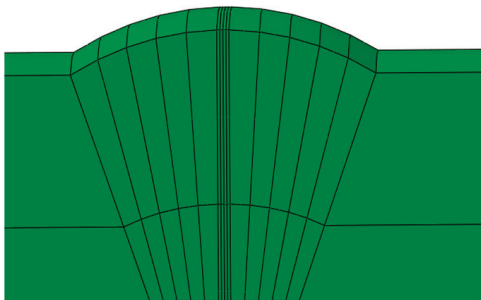
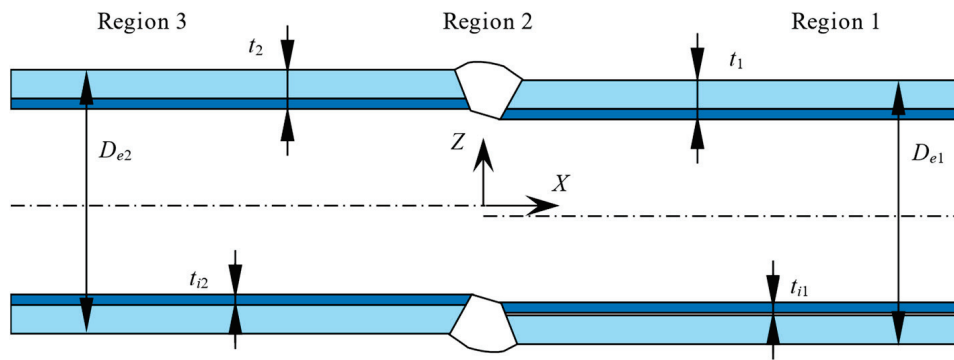
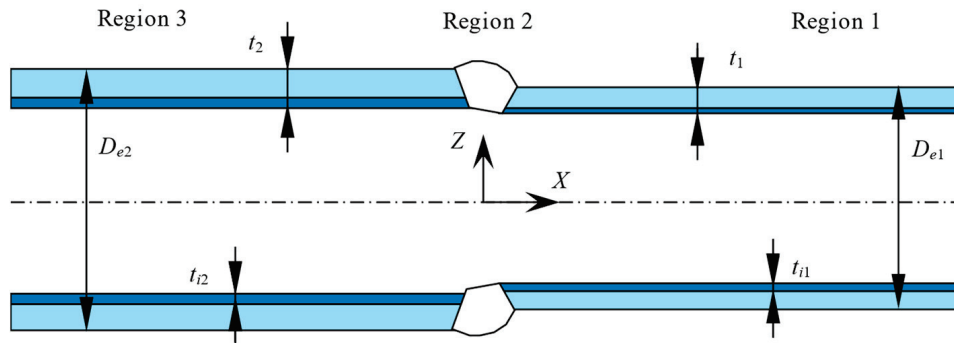


Figure A3. Typical FE meshes at the girth weld.

Then, the coordinate transformation was adopted to generate FE models of bi-layer pipelines with misalignment at girth welds from the FE model of aligned bi-layer pipelines. The coordinate transformation formulas presented below are essential for adapting aligned pipeline models to accommodate misalignments, reflecting more realistic and complex real-world scenarios. This addition enhances the applicability of the finite element method in pipeline structural and stress analysis, allowing for more accurate simulations and analyses of pipelines under various conditions, including those that are not perfectly aligned. Figure A4a,b illustrates the coordination transformation of bi-layer pipe, which has centerline offset with equal thickness and diameters and bi-layer pipe, which has centerline offset with varying thickness but aligned inner surfaces, respectively.



(a) Bi-layer pipe having centerline offset with equal thickness and diameters



(b) Bi-layer pipe having centerline offset with varying thickness but aligned inner surfaces

Figure A4. Coordinate transformation of typical bi-layer pipes with misalignment.

For bi-layer pipe with a centerline offset with equal thickness and diameters (Figure A4a), the transformation from an aligned (x -, y -, z - coordinate system) to a misaligned configuration (X -, Y -, Z - coordinate system) is executed using the specified expression as follows.

$$\text{Region 1} \begin{cases} X = x \\ Y = y \\ Z = z - e \end{cases} \quad (\text{A15a})$$

$$\text{Region 2} \begin{cases} X = x \\ Y = y \\ Z = z - e_{\text{eff}} \end{cases} \quad (\text{A15b})$$

$$\text{Region 3} \quad \begin{cases} X = x \\ Y = y \\ Z = z \end{cases} \quad (\text{A15c})$$

where

$$e_{\text{eff}} = \left(\frac{2x + L_e + L_v}{2L_v} \right) e \quad (\text{A16})$$

$$L_v = L_i + \frac{[2(r + t_2) - D_{e2}](L_e - L_i)}{2t_2} \quad (\text{A17})$$

$$r = \sqrt{y^2 + z^2} \quad (\text{A18})$$

For a bi-layer pipe with a centerline offset with varying thickness but aligned inner surfaces (Figure A4b), there are two scenarios: $r \leq r_c$ and $r > r_c$ with $r_c = D_{e2}/2 - (t_2 - t_{i2})$. Here, t_{i2} represents the clad layer thickness, as shown in Figure A4b. The coordinate transformation for these two cases is summarized through specific expressions as follows.

For the first scenario, $r \leq r_c$:

$$\text{Region 1} \quad \begin{cases} X = x \\ Y = (r - e_{v1})\sin \theta \\ Z = (r - e_{v1})\cos \theta \end{cases} \quad (\text{A19a})$$

$$\text{Region 2} \quad \begin{cases} X = x \\ Y = (r - e_{v1,\text{eff}})\sin \theta \\ Z = (r - e_{v1,\text{eff}})\cos \theta \end{cases} \quad (\text{A19b})$$

$$\text{Region 3} \quad \begin{cases} X = x \\ Y = y \\ Z = z \end{cases} \quad (\text{A19c})$$

where

$$e_{v1} = (t_{i2} - t_{i1}) \left[\frac{r - (D_{e2}/2 - t_2)}{t_{i2}} \right] \quad (\text{A20})$$

$$e_{v1,\text{eff}} = \left(\frac{2x + L_e + L_v}{2L_v} \right) e_{v1} \quad (\text{A21})$$

For the second scenario, $r > r_c$:

$$\text{Region 1} \quad \begin{cases} X = x \\ Y = (r - e_{v2})\sin \theta \\ Z = (r - e_{v2})\cos \theta \end{cases} \quad (\text{A22a})$$

$$\text{Region 2} \quad \begin{cases} X = x \\ Y = (r - e_{v2,\text{eff}})\sin \theta \\ Z = (r - e_{v2,\text{eff}})\cos \theta \end{cases} \quad (\text{A22b})$$

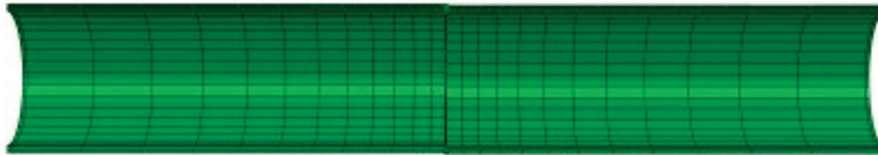
$$\text{Region 3} \quad \begin{cases} X = x \\ Y = y \\ Z = z \end{cases} \quad (\text{A22c})$$

where

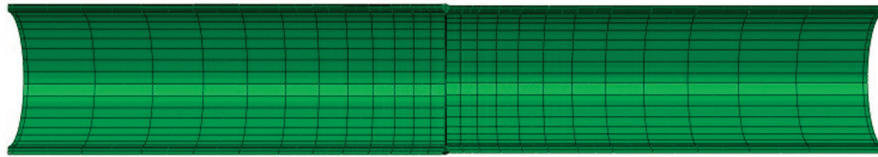
$$e_{v2} = (t_{i2} - t_{i1}) + \left(1 - \frac{t_1 - t_{i1}}{t_2 - t_{i2}}\right)(r - r_c) \quad (\text{A23})$$

$$e_{v2,\text{eff}} = \left(\frac{2x + L_e + L_v}{2L_v}\right)e_{v2} \quad (\text{A24})$$

Consequently, the FE models of the two typical bi-layer pipes with misalignment can be generated, which can be seen in Figure A5.



(a) Bi-layer pipe having centerline offset with equal thickness and diameters



(b) Bi-layer pipe having centerline offset with varying thickness but aligned inner surfaces

Figure A5. FE models of typical bi-layer pipes with misalignment.

The FE models in this study were generated using the ABAQUS software V2021. Given the consideration of large deformations, the C3D20R element is used for the numerical simulation. It is difficult to determine the mesh density directly; hence, the convergence test is recommended to determine the number of elements in the FE models. One can gradually increase the mesh density until the simulation results converge. In this study, the number of elements increases to 1×10^4 to make the results converge. In addition, the axial force is applied at one end of the pipe, and the fixed boundary condition is applied on the other.

References

1. Marshall, P.W. A review of stress concentration factors in tubular connections. In *Report CE-32*; Shell: Houston, TX, USA, 1978.
2. Zavvar, E.; Henneberg, J.; Soares, C.G. Stress concentration factors in FRP-reinforced tubular DKT joints under axial loads. *Mar. Struct.* **2023**, *90*, 103429. [CrossRef]
3. Bao, S.; Wang, W.; Li, X.; Zhou, J. Stress concentration factor distribution formulas for three-planar tubular Y-joints under axial force. *Ocean. Eng.* **2022**, *265*, 112687. [CrossRef]
4. Nassiraei, H.; Rezadoost, P. Stress concentration factors in tubular T-joints reinforced with external ring under in-plane bending moment. *Ocean. Eng.* **2022**, *266*, 112551. [CrossRef]
5. Sofiani, F.M.; Chaudhuri, S.; Elahi, S.A.; Hectors, K.; De Waele, W. Quantitative analysis of the correlation between geometric parameters of pits and stress concentration factors for a plate subject to uniaxial tensile stress. *Theor. Appl. Fract. Mech.* **2023**, *127*, 104081. [CrossRef]
6. Brabin, T.A.; Christopher, T.; Rao, B.N. Finite element analysis of cylindrical pressure vessels having a misalignment in a circumferential joint. *Int. J. Press. Vessel. Pip.* **2010**, *87*, 197–201. [CrossRef]
7. Maddox, S.J. *Fitness for Purpose Assessment of Misalignment in Transverse Butt Welds Subjected to Fatigue Loading*; IIW Document XIII-1180; International Institute of Welding: Genoa, Italy, 1985.
8. Efthymiou, M. Development of SCF formulae and generalised influence functions for use in fatigue analysis. In *Recent Developments in Tubular Joint Technology*; OTJ'88: London, UK, 1988.
9. Smedley, S.; Fischer, P. Stress concentration factors for ring-stiffened tubular joints. In *Proceedings of the 1st International Offshore and Polar Engineering Conference*, Edinburgh, UK, 11–16 August 1991.
10. Lotsberg, I. Stress concentration factors at circumferential welds in tubulars. *Mar. Struct.* **1998**, *11*, 203–230. [CrossRef]

11. Lotsberg, I. Stress concentration factors at welds in pipelines and tanks subjected to internal pressure and axial force. *Mar. Struct.* **2008**, *21*, 138–159. [CrossRef]
12. Lotsberg, I. Stress concentrations due to misalignment at butt welds in plated structures and at girth welds in tubulars. *Int. J. Fatigue* **2009**, *31*, 1337–1345. [CrossRef]
13. Timoshenko, S.P.; Woinowsky-Krieger, S. *Theory of Plates and Shells*, 2nd ed.; McGraw-Hill Book Company, Inc.: New York, NY, USA, 1959.
14. DNV-RP-C203; Fatigue Design of Offshore Steel Structures. Det Norske Veritas AS: New York, NY, USA, 2011.
15. BS7910-Amendment 1; Guide to Methods for Assessing the Acceptability of Flaws in Metallic Structures. British Standards Institution: Jersey, UK, 2013.
16. Zhao, H.S.; Lie, S.T.; Zhang, Y. Stress intensity factors for semi-elliptical surface cracks in plate-to-plate butt welds with parallel misalignment of same thickness. *Mar. Struct.* **2017**, *53*, 148–163. [CrossRef]
17. Yi, D.; Xiao, Z.M.; Idapalapati, S.; Kumar, S.B. Fracture analysis of girth welded pipelines with 3D embedded cracks subjected to biaxial loading conditions. *Eng. Fract. Mech.* **2012**, *96*, 570–587. [CrossRef]
18. Chang, Q.; Cao, Y.G.; Zhen, Y.; Wu, G.; Li, F.G. Study on the effect of loading conditions on the fracture behavior of pipeline with girth weld. *Int. J. Press. Vessel. Pip.* **2023**, *203*, 104940. [CrossRef]
19. Cao, Y.G.; Chang, Q.; Zhen, Y. Numerical simulation of fracture behavior for the pipeline with girth weld under axial load. *Eng. Fail. Anal.* **2022**, *136*, 106221. [CrossRef]
20. Gong, B.M.; Wan, X.B.; Huo, X.T.; Liu, Y.; Deng, C.Y.; Wang, D.P.; Dai, L.S. Development of a strain-based fracture assessment approach for X80 steel pipe welded girth by modified Mohr-Coulomb model. *Int. J. Press. Vessel. Pip.* **2023**, *206*, 105056. [CrossRef]
21. Zhao, H.S.; Lie, S.T.; Zhang, Y. Elastic-plastic fracture analyses for misaligned clad pipeline containing a canoe shape surface crack subjected to large plastic deformation. *Ocean. Eng.* **2017**, *146*, 87–100. [CrossRef]
22. Zhang, Y.H.; Shuai, J.; Lv, Z.Y.; Zhang, T.Y. Study on fracture behavior of pipeline girth weld based on CTOD-Am method and Gurson ductile damage model. *Theor. Appl. Fract. Mech.* **2023**, *123*, 103692. [CrossRef]
23. Zhao, H.S.; Hu, Y.Y.; Bi, C.W.; Li, X. Numerical study on hydrodynamic behaviors of and flow field around UHMWPE plane nets. *Aquac. Eng.* **2024**, *106*, 102397. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Research on a Real-Time Prediction Method of Hull Girder Loads Based on Different Recurrent Neural Network Models

Qiang Wang ¹, Lihong Wu ^{1,*}, Chenfeng Li ², Xin Chang ¹ and Boran Zhang ³

¹ College of Naval Architecture and Ocean Engineering, Dalian Maritime University, Dalian 116026, China; wangqiang5120@dlmu.edu.cn (Q.W.); changxin_heu@outlook.com (X.C.)

² College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China; lichenfeng@hrbeu.edu.cn

³ Department of Infrastructure Engineering, The University of Melbourne, Melbourne, VIC 3010, Australia; boran@student.unimelb.edu.au

* Correspondence: wlh@dlmu.edu.cn

Abstract: Real-time prediction of hull girder loads is of great significance for the safety of ship structures. Some scholars have used neural network technology to investigate hull girder load real-time prediction methods based on motion monitoring data. With the development of deep learning technology, a variety of recurrent neural networks have been proposed; however, there is still a lack of systematic comparative analysis on the prediction performance of different networks. In addition, the real motion monitoring data inevitably contains noise, and the effect of data noise has not been fully considered in previous studies. In this paper, four different recurrent neural network models are comparatively investigated, and the effect of different levels of noise on the prediction accuracy of various load components is systematically analyzed. It is found that the GRU network is suitable for predicting the torsional moment and horizontal bending moment, and the LSTM network is suitable for predicting the vertical bending moment. Although filtering has been applied to the original noise data, the prediction accuracy still decreased as the noise level increased. The prediction accuracy of the vertical bending moment and horizontal bending moment is higher than that of the torsional moment.

Keywords: ship structure; hull girder loads; recurrent neural network; ship motion; noise level

1. Introduction

The hull girder loads are an important factor affecting the overall structural safety of ships, which is a key focus in the design and use of ship structures [1,2]. Real-time prediction of hull girder load data will be of great help to the captain's navigation decision-making, especially in high-sea conditions when the hull girder loads are large and there is a high risk of failure in the ship's structure.

The Hull Response Monitoring System (HRMS) is a system that measures and displays ship structure stress, ship structure strain, and ship motion data. Many scholars have established a real-time prediction method for hull girder loads based on real-time strain monitoring data by utilizing the mechanical relationship between hull girder loads and structural strain [3–8]. Strain sensors on the hull structure may experience data drift and other issues during long-term use, which can lead to incorrect reporting of the hull girder loads. Compared with the strain gauge data, the ship motions are more accessible to obtain and more stable. For ships without HRMS installed, the hull girder loads can be indirectly predicted through motion monitoring data. For ships that have already been installed with HRMS, the hull girder loads obtained by using motion data and strain data can be compared to ensure the accuracy of hull girder load measurements further.

Xu [9] developed a methodology for estimating wave-induced vertical bending moments from heave and pitch measurements based on artificial neural networks. The artificial

neural networks were trained by using the auto- and cross-correlation functions of the bending moment and the ship motions. The prediction model has been validated through a comparison of experimental data. Moreira and Guedes Soares [10] presented a neural network model for the estimation of hull bending moment and shear force of ships in waves. The ship motion and hull girder loads were simulated by using a time domain strip method [11], and then a reasonable network model was established to estimate the instantaneous bending moment and shear force by analyzing the effect of different numbers of neurons. Xu et al. [12] also utilized an improved BP neural network to investigate the prediction method of sectional shear force and bending moment based on ship motion data. Previous research tends to focus more on utilizing networks to train the correlation between motion and hull girder loads. However, due to the memory effect of waves, ship motion and hull girder loads exhibit strong temporal correlations.

With the advancement of networks, recurrent neural networks (RNNs) with long-term memory effects are increasingly being utilized in the fields of ship and ocean engineering. Liu et al. [13] presented a model for the reconstruction and prediction of global whipping responses on a large cruise ship based on Long Short-Term Memory (LSTM) neural networks. Su et al. [14] introduced a real-time algorithm for predicting ship vertical acceleration using the LSTM and GRU models. Based on the database originating from a commercial professional simulator, Li et al. [15] predicted short-term ship roll motion by leveraging the encoder–decoder structure of Bidirectional Long Short-Term Memory Networks (Bi-LSTM) with teacher forcing. Fan et al. [16] combined the different empirical mode decomposition methods with LSTM networks to effectively predict short-term offshore structure motions. Liu et al. [17] proposed an input vector space optimization for the LSTM deep learning model in real-time prediction of ship motions. Xie et al. [18] introduced a mooring line tension model based on the GRU network, which accurately predicts mooring line tension using wave elevation components or float motion responses.

Many scholars use recurrent neural networks to solve time-series problems with temporal correlations in the ship and ocean engineering field. However, the performance of different RNNs in real-time prediction of the hull girder loads still lacks systematic research. In addition, real-time monitoring data of ship motion, as input data for the real-time prediction model of the hull girder load, inevitably contains certain noise components. However, there is a lack of detailed analysis of the impact of noise on motion input data in previous studies.

In this paper, the real-time prediction method of hull girder loads is investigated by using several different recurrent neural network models. The performance of different recurrent neural networks is systematically compared and analyzed. The method of controlling variables is used for hyperparameter optimization. Taking into account the characteristic that the ship motion data contains noise, a new prediction procedure combining filtering techniques to reduce the influence of noise is established. In addition, the effect of different levels of noise on the prediction accuracy of hull girder loads is systematically analyzed.

2. Hull Girder Loads Prediction Model

2.1. Basic Concepts of RNN

Simple recurrent networks can theoretically establish dependencies between states with long-time intervals. The gradient explosion or gradient vanishing problem restricts the learning of only short-term dependencies. The Long Short-Term Memory Network (LSTM) is a variant of recurrent neural networks that can effectively solve the problems of gradient explosion or vanishing encountered in simple RNNs. The LSTM model calculates the gradient through the backpropagation through time (BPTT) algorithm and then adjusts the model weight and bias parameters until convergence. The sensitivity of the model to time is reflected in the three gating units (input gate, forgetting gate, and output gate) that control the information flow. Through these three gating units, information can be selectively remembered and forgotten [19].

The Gated Recurrent Unit (GRU) network represents a simpler form of the recurrent neural network compared to the LSTM network. It incorporates a gating mechanism to regulate the update of information. Unlike LSTM, GRU omits an additional memory unit and employs an update gate to determine the amount of historical state information to retain in the current state, as well as how much new information from the candidate set should be considered for inclusion [20].

Increasing the depth of recurrent neural networks enhances model performance, commonly achieved by stacking multiple recurrent networks, forming a Stacked Recurrent Neural Network (SRNN). A Bidirectional Recurrent Neural Network (BI-RNN) comprises two layers of recurrent neural networks with identical inputs, differing only in the direction of information flow. In conjunction with the LSTM neural network and the GRU neural network, the bidirectional recurrent neural network may encompass a bidirectional long short-term memory (BI-LSTM) neural network [15] and a bidirectional gated recurrent unit (BI-GRU) neural network [21].

2.2. Input and Output Data

In this study, the RNN models are developed to predict the hull girder loads by the known ship motion responses. A container ship is chosen as the research object, and the numerical simulation of the ship is conducted in the time domain. The specific parameters are shown in the following Section 3.1, taking the ship's heave motion, pitch motion, and roll motion as input, and the vertical bending moment, horizontal bending moment, and torsional moment at the midship section as output. After giving the complete input and output time histories of each operating sea state in the data of RNN models, the data are divided into the training and validating sets, which are respectively used for training the RNN models and validating the accuracy of the predicting results from the trained RNN models. In addition, the validating sets should be independent of the training sets; that is, validating sets do not participate in the training process of the neural network model.

The motion data of the ship obtained by the hull response monitoring systems (HRMS) are usually accompanied by a significant amount of noise. Numerical motion data obtained from simulations are artificially augmented with numerical noise, forming testing sets with motion input data that includes noise. The testing sets are designed with inputs of data containing various levels of noise to test the accuracy of the predicted results.

2.3. Data Pre-Processing

The normalized data can enhance the convergence speed of the network model and improve the prediction performance. Input and output data often differ significantly in their numerical values and dimensions. Normalization helps to eliminate these differences and achieve consistency in the data, facilitating more effective training of the model.

$$x' = \frac{x - x_{\min}}{x_{\max} - x_{\min}} \quad (1)$$

where x' is the normalized input or output data, x is the original data, and $x_{\max}$ and $x_{\min}$ are the maximum and minimum value in the sequence, respectively. To ensure a meaningful comparison with the actual hull girder load sequence, it is necessary to apply an inverse normalization to the predicted results.

2.4. Evaluation Criterion

Evaluation criteria are crucial parameters for evaluating the prediction performance of the model. Root Mean Square Error (RMSE), Mean Absolute Percentage Error (MAPE), and Pearson's correlation coefficient (ρ) are selected.

RMSE represents the standard deviation of the residuals between the predicted and actual values, providing a measure of error discretization. It is defined as

$$RMSE = \sqrt{\frac{1}{n} \sum_{i=1}^n (y_i^{origin} - y_i^{pred})^2} \quad (2)$$

where y_i^{origin} is the actual value, y_i^{pred} is the predicted value, n represents the quantity of data, and the range of the *RMSE* value is $[0, +\infty]$, which is equal to 0 if the predicted value exactly matches the true value; the larger the error, the larger the value.

MAPE is a percentage representation of the *MAE*, which is the average of the absolute value of the error:

$$MAPE = \frac{100\%}{n} \sum_{i=1}^n \left| \frac{y_i^{origin} - y_i^{pred}}{y_i^{origin}} \right| \quad (3)$$

where y_i^{origin} is the actual value, y_i^{pred} is the predicted value, and n represents the quantity of data. The range of *MAPE* is $[0, +\infty]$ and a *MAPE* of 0% indicates a perfect model; the closer the value is to 0, the better the model's predictive accuracy.

Pearson's correlation coefficient quantifies the linear relationship between two random variables. The concept of correlation coefficient is introduced:

$$\rho = Corr(X, Y) = \frac{Cov(X, Y)}{\sigma_X \cdot \sigma_Y} \quad (4)$$

Here, σ_X , σ_Y represent the standard deviations of variables X , Y , respectively. $Cov(X, Y)$ is the covariance of X , Y . The closer the correlation coefficient (ρ) is to 1, the stronger the correlation in the model and the more accurate the model prediction.

3. Physics-Based Numerical Simulation

3.1. Simulation Model

A large container ship is selected as the object of this paper. Table 1 shows the main parameters of the container ship. Motion simulation in irregular waves is conducted using the commercial software Wasim to generate databases of ship motion responses and hull girder loads. Wasim is a three-dimensional hydrodynamic analysis software based on potential flow theory in the time domain; it can predict ship motion responses in waves with the Rankine panel method and considers many nonlinear factors. The fluid dynamic pressure field can be generated based on Bernoulli's equation. The d'Alembert principle is used to solve the sectional-induced loads of the ship in waves.

Table 1. Container ship's principal dimensions.

Length	Width	Depth	Structural Draught	Speed	Block Coefficient
278 m	32.2 m	23.3 m	13 m	23 kn	0.7

Figure 1 illustrates the wet surface mesh. The Pierson–Moskowitz (PM) spectrum is chosen as a wave spectrum to describe irregular waves. Irregular waves are represented using 200 sub-waves. Eight percent of the critical damping is applied to roll motion. The control of gravity is assigned as (124.63 m, 0, 10.22 m). A non-linear solver is employed for the 3 h time domain simulation.

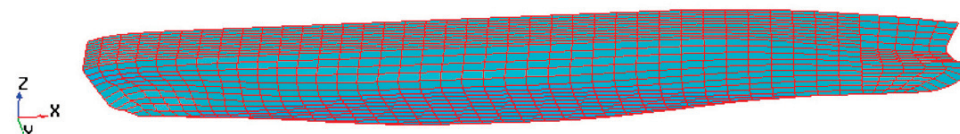


Figure 1. Wet surface mesh.

3.2. Simulation Load Case

The databases of ship motion responses and hull girder loads under different short-term sea conditions are established to train the network models. Parameters related to short-term sea conditions include significant wave height H_s , average zero-crossing period T_z , wave direction β , and ship speed v . When the sea condition parameters encountered during the ship's voyage are the same as those in the training database, the data have strong relevance, which may lead to higher forecast accuracy.

Considering that the actual sea conditions experienced by the ship may not be consistent with the sea conditions in the training set, the load case in Table 2 is designed to verify the predictive ability of the neural network model. The sea state parameters encountered by the ship are assumed to be $H_s = 11$ m, $T_z = 11$ s, $\beta = 135^\circ$, and $v = 7.5$ kn, as indicated in Table 2 for the testing database. Sixteen sets of short-term sea conditions for numerical simulation are assumed as the training and validating databases. In subsequent practical applications, it is only necessary to further increase the number of samples in the training set to obtain a neural network model with a wider applicability.

Table 2. Loadcase parameters of B_MULTI.

Load Case	Databases	Significant Wave Height H_s (m)	Zero-Crossing Periods T_z (s)	Wave Direction β ($^\circ$)	Velocity v (kn)
B_MULTI	Training and validating	10.5	10.5	120	5
		10.5	10.5	120	10
		10.5	10.5	150	5
		10.5	10.5	150	10
		10.5	11.5	120	5
		10.5	11.5	120	10
		10.5	11.5	150	5
		10.5	11.5	150	10
		11.5	10.5	120	5
		11.5	10.5	120	10
		11.5	10.5	150	5
		11.5	10.5	150	10
		11.5	11.5	120	5
		11.5	11.5	120	10
		11.5	11.5	150	5
		11.5	11.5	150	10
	Testing	11	11	135	7.5

4. Results and Discussion

4.1. Determination of Network Structures of Different RNN Models

The training process of the neural network is to find the suitable fitting function in the hypothesis space on a certain number of training data. The control variable method is the common method to find the optimal model by adjusting the single variable and maintaining other variables unchanged [16,22,23]. In this section, the influence of window length, optimizers, layer number, and neuron number on the accuracy of hull girder loads prediction models will be studied in turn.

4.1.1. Determination of Window Length

The variation in window length is one of the key factors affecting the prediction accuracy of the model. When the window length is set to 'a', its physical meaning is that the motion data from the previous 'a' time steps are used as inputs, and the ship hull girder loads at the 'a'th time step are used as the output. By sliding the window, multiple databases of motion inputs and ship hull girder load outputs are obtained.

For the torsional moment (TM) prediction model, to investigate the impact of different window lengths on network prediction performance, window lengths of 50, 100, 150, and

200 are analyzed. A single-layer neural network with 32 neurons and the ‘adam’ optimizer is chosen, and four networks—LSTM, GRU, BI-LSTM, and BI-GRU—are separately trained.

The GRU network model exhibits the highest prediction accuracy, followed by the BI-GRU network model, whereas the LSTM and BI-LSTM network models demonstrate lower prediction accuracy and larger errors, as shown in Figures 2 and 3. For the GRU network, a window length of 150 yields $\rho = 0.772$, and $\text{RMSE} = 0.051$, significantly outperforming the window length of 50 and slightly outperforming the window lengths of 100 and 200. The window lengths of 150, 100, and 150 are the optimal window lengths for the LSTM, BI-LSTM, and BI-GRU networks, respectively.

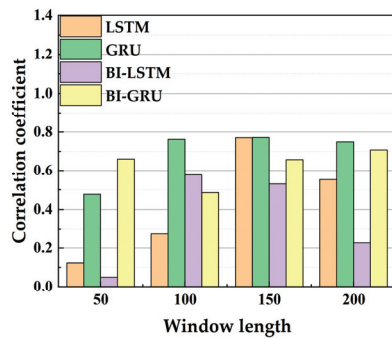


Figure 2. Correlation coefficient (TM model).

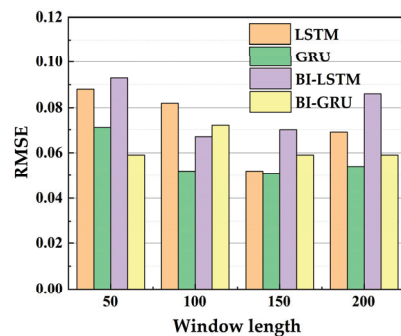


Figure 3. RMSE (TM model).

For the vertical bending moment (VBM) prediction model, a single-layer neural network with 32 neurons using the ‘adam’ optimizer was chosen. The evaluation index value for a window length of 150 is slightly higher than that for the other three lengths, indicating that the accuracy of the four neural network models is not significantly affected by changes in window length, as shown in Figures 4 and 5. For a window length of 150, the precision (ρ) of all four networks exceeds 0.9, and the RMSE is less than 0.037. Specifically, the LSTM network model achieves the highest precision of $\rho = 0.935$ and $\text{RMSE} = 0.032$ among all networks. The window lengths of 150, 100, and 150 are the optimal window lengths for the GRU, BI-LSTM, and BI-GRU networks, respectively.

For the horizontal bending moment (HBM) prediction model, a single-layer neural network with 64 neurons using the ‘adam’ optimizer is employed. The GRU network model exhibits the highest prediction accuracy, followed by the BI-GRU network model, while the prediction accuracy of the LSTM and BI-LSTM network models is slightly lower, as shown in Figures 6 and 7. Among GRU networks, at a window length of 150, $\rho = 0.837$ and $\text{RMSE} = 0.057$, demonstrating notably superior performance compared to a window length of 100, and marginally better performance compared to window lengths of 50 and 200. The window lengths of 200, 200, and 200 are the optimal window lengths for the LSTM, BI-LSTM, and BI-GRU networks, respectively.

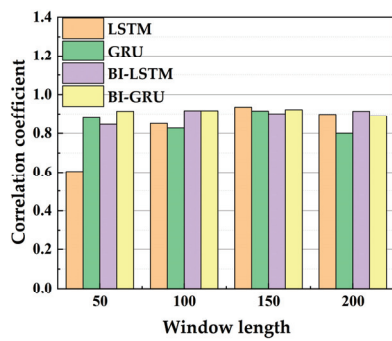


Figure 4. Correlation coefficient (VBM model).

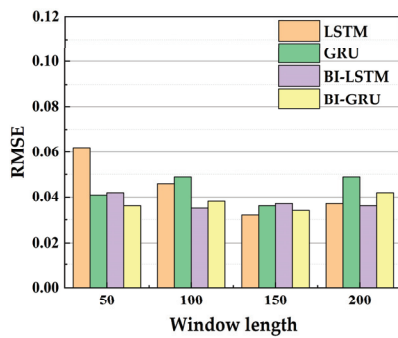


Figure 5. RMSE (VBM model).

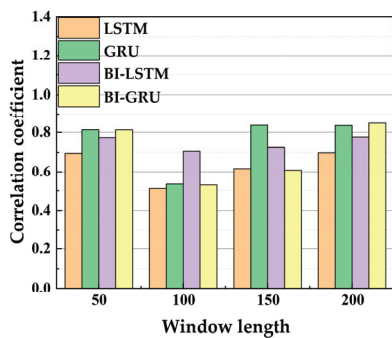


Figure 6. Correlation coefficient (HBM model).

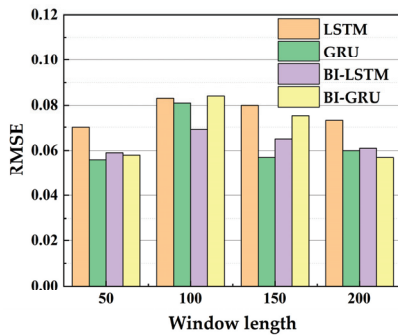


Figure 7. RMSE (HBM model).

4.1.2. Determination of Optimizers

The variation in optimizers is another one of the key factors affecting the prediction accuracy of the model. The role of the optimizer in the backpropagation process of deep learning is to guide the parameters of the loss function to update in the correct direction

with an appropriate magnitude, allowing the loss function to gradually approach or achieve its minimum value.

For the TM prediction model, to investigate the impact of different optimizers on network prediction performance, optimizers of SGD, adam, RMSprop, and Nadam are analyzed. A single-layer neural network with 32 neurons and a window length of 150 is chosen, and four networks—LSTM, GRU, BI-LSTM, and BI-GRU—are separately trained.

Figures 8 and 9 illustrate that the adam and Nadam optimizers achieve relatively high prediction accuracy across all four neural network models, surpassing significantly the accuracy obtained with SGD and RMSprop optimizers. Among the four models, the traditional optimizer SGD exhibits the lowest prediction accuracy. Among the four network models, the GRU network demonstrates the highest prediction accuracy, followed by the LSTM network. Regarding the GRU network, the adam optimizer yields significantly better prediction accuracy compared to RMSprop and slightly outperforms the Nadam optimizer, as shown in Figure 10. The adam optimizer exhibits results closest to the actual values; hence it is selected for constructing the GRU network. The optimizer of adam is also the optimal optimizer for LSTM and BI-GRU. Nadam is the optimal optimizer for the BI-LSTM network.

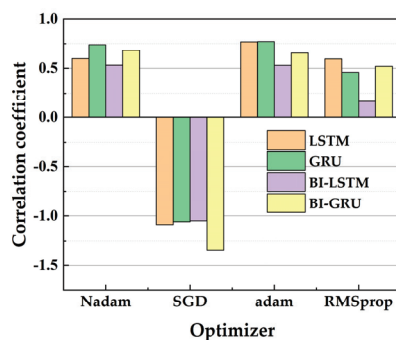


Figure 8. Correlation coefficient (TM model).

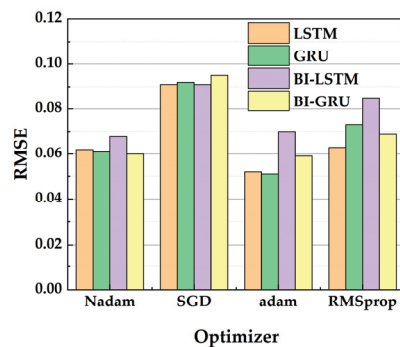


Figure 9. RMSE (TM model).

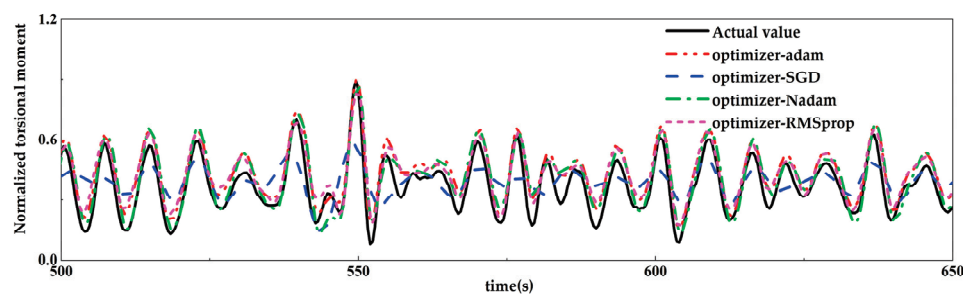


Figure 10. Prediction results under different optimizers (GRU network).

For the VBM prediction model, a single-layer neural network with 32 neurons and a window length of 150 is used. As shown in Figures 11 and 12, it is evident that the adam,

Nadam, and RMSprop optimizers have relatively higher prediction accuracy than the SGD optimizers. The LSTM network model demonstrates the highest prediction accuracy among the four network models, followed by the BI-GRU network model. In LSTM networks, the prediction accuracy achieved with the adam optimizer is comparable to that obtained with RMSprop and Nadam. Figure 13 illustrates the prediction results obtained with different optimizers in the LSTM network. The adam optimizer yields predictions closest to the true values; thus, it is selected for constructing the LSTM network. The optimizer of adam is also the optimal optimizer for GRU and BI-GRU. RMSprop is the optimal optimizer for the BI-LSTM network.

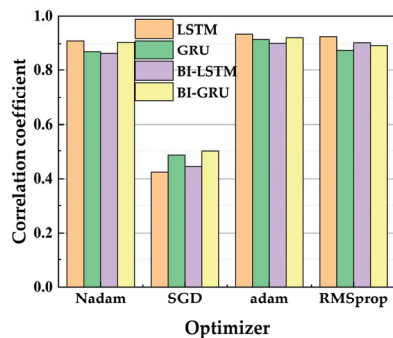


Figure 11. Correlation coefficient (VBM model).

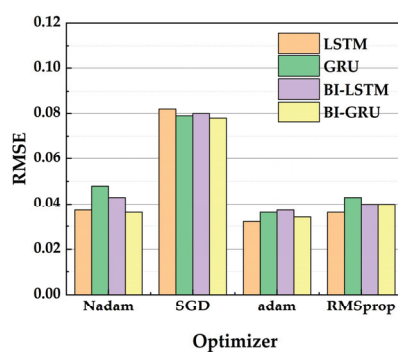


Figure 12. RMSE (VBM model).

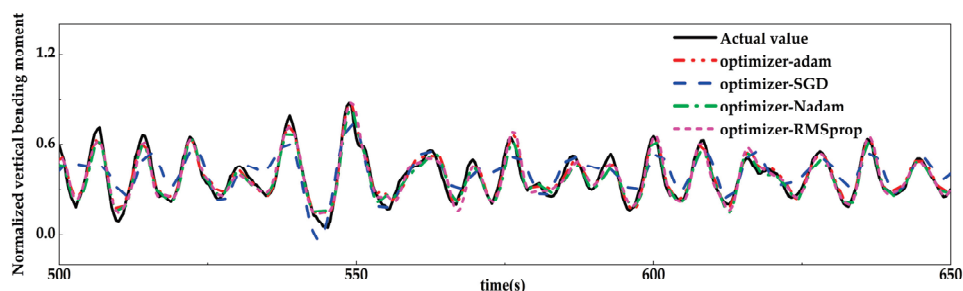


Figure 13. Prediction results under different optimizers (LSTM network).

For the HBM prediction model, a single-layer neural network with 64 neurons and a window length of 150 is used. As shown in Figures 14 and 15, the adam optimizer consistently yields relatively high prediction accuracy across all four neural network models, slightly outperforming the Nadam optimizer. Conversely, the RMSprop optimizer exhibits relatively low prediction accuracy, while the SGD optimizer, the most traditional choice, performs the worst among the four models. Among the four network models, the GRU network model achieves the highest prediction accuracy, followed by the BI-LSTM network model. In GRU networks, the adam optimizer yields slightly higher prediction accuracy compared to the Nadam and RMSprop optimizers. Figure 16 illustrates the

prediction results obtained with different optimizers in the GRU network. The adam optimizer yields predictions closest to the true values; thus, it is selected for constructing the GRU network. The optimizer of adam is also the optimal optimizer for LSTM and BI-LSTM. Nadam is the optimal optimizer for the BI-GRU network.

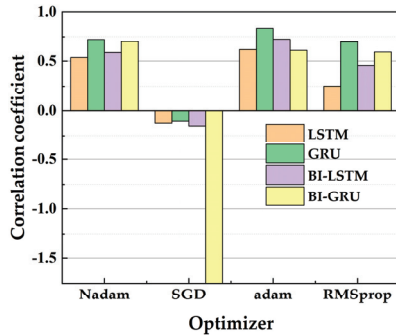


Figure 14. Correlation coefficient (HBM model).

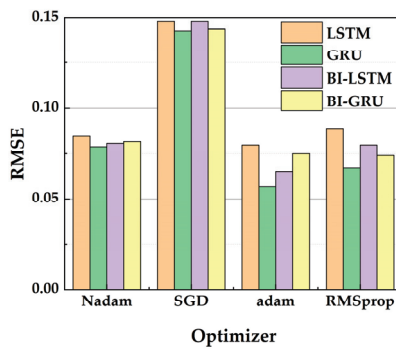


Figure 15. RMSE (HBM model).

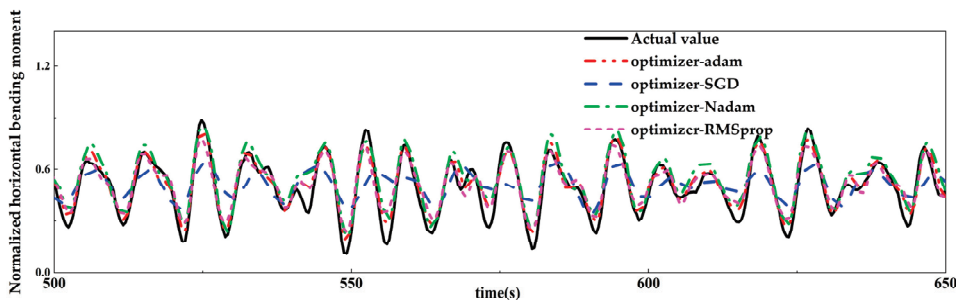


Figure 16. Prediction results under different optimizers (GRU network).

4.1.3. Coupling Determination of the Layer and Neuron Number

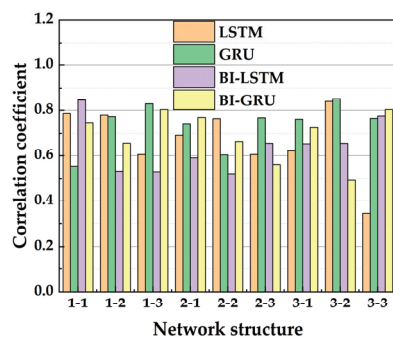
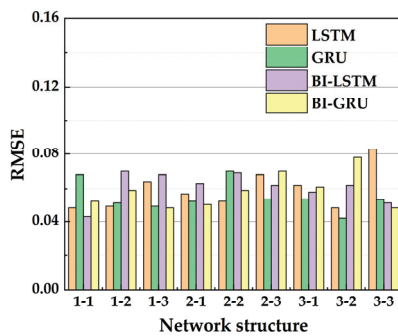
The number of neurons and the depth of the neural network are the key factors affecting the prediction accuracy of the model. Considering the coupled impact of the number of network layers and the number of neurons on the network's prediction accuracy, the most suitable network structure is identified.

For the TM prediction model, to investigate the impact of different numbers of network layers and neurons on network prediction performance, nine types of network structure were analyzed. A window length of 150 and the 'adam' optimizer are chosen. Table 3 illustrates the number of network layers and neurons.

Table 3. Number of network layers and number of neurons (TM model).

Item	1-1	1-2	1-3	2-1	2-2	2-3	3-1	3-2	3-3
Network Layers	1	1	1	2	2	2	3	3	3
Number of neurons	16	32	64	16-8	32-16	32-32	16-32-16	32-32-32	64-32-16

Figures 17 and 18 illustrate that for a single layer (layer = 1), the BI-LSTM network achieves the highest prediction accuracy with $\rho = 0.846$ and RMSE = 0.043; for two layers (layer = 2), the GRU network exhibits the highest prediction accuracy with $\rho = 0.767$ and RMSE = 0.054; and for three layers (layer = 3), the GRU network attains the highest prediction accuracy with $\rho = 0.853$ and RMSE = 0.042. For a single layer (layer = 1), the prediction accuracy of the network rises with an increasing number of neurons; while for three layers (layer = 3) with 32 neurons each, the network achieves the highest prediction accuracy. Figure 19 illustrates the prediction results of the GRU network with different numbers of hidden layers.

**Figure 17.** Correlation coefficient (TM model).**Figure 18.** RMSE (TM model).

In the case of a three-layer GRU network, changing the number of neurons per layer to 64, 32, and 16 results in lower prediction accuracy compared to when the number of neurons remains consistent at 32 for all layers. This suggests that excessive neurons may lead to overfitting. Moreover, having varying numbers of neurons for the same number of layers does not necessarily improve performance; hence, thorough parameter discussion is crucial to determine the optimal settings. The three-layer networks with 32 neurons each and the single-layer networks with 16 neurons and 64 neurons are the optimal network structures for LSTM, BI-LSTM, and BI-GRU networks, respectively.

For the VBM prediction model, four different numbers of hidden layers and various numbers of neurons are chosen to analyze the influence of coupling parameters. The window length is set to 150, and the 'adam' optimizer is employed. Table 4 illustrates the number of network layers and neurons.

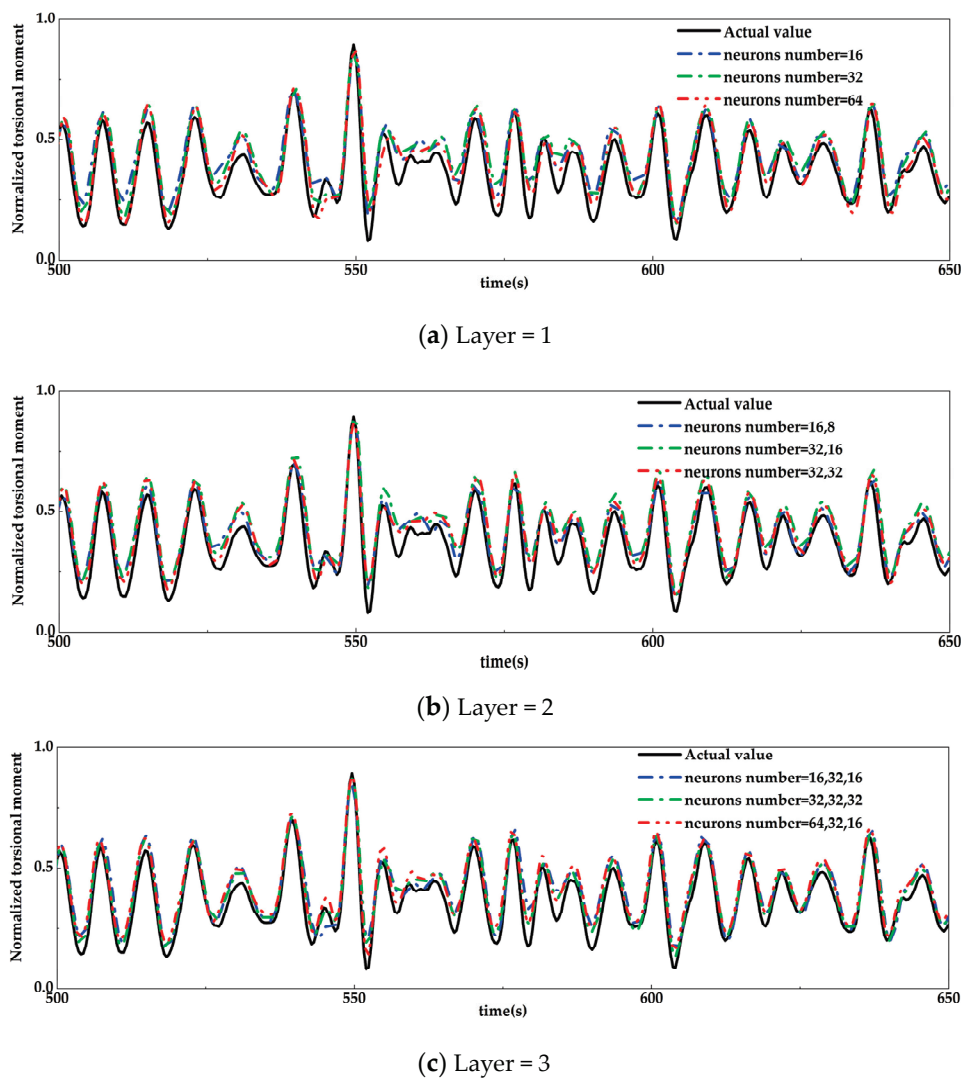


Figure 19. Prediction results under different network structures (GRU network).

Table 4. Number of network layers and number of neurons (VBM model).

Item	1-1	1-2	1-3	2-1	2-2	3-1	4-1
Network Layers	1	1	1	2	2	3	4
Number of neurons	8	16	32	16-8	32-16	16-32-16	16-32-32-16

Figures 20 and 21 illustrate that when the number of layers is set to 1, the prediction accuracy of all networks increases as the number of neurons increases. Figure 22 illustrates the prediction results obtained with different numbers of neurons in a single-layer LSTM network. When the number of neurons is set to 32, all four networks achieve a prediction accuracy of over 0.9. The highest prediction accuracy is achieved with 32 neurons in the LSTM network. When the number of layers is 2, 3, and 4, despite the gradual increase in the number of layers, the number of neurons also increases. However, the prediction accuracy of all four networks remains around 0.8–0.9. The networks show insensitivity to changes in the number of layers and neurons, but increasing the number of layers and neurons prolongs the training time. The single-layer LSTM network model demonstrates relatively high prediction accuracy and training efficiency. The two-layer networks with 16-8 neurons, the three-layer networks with 16-32-16 neurons, and the one-layer network with 32 neurons are the optimal network structures for GRU, BI-LSTM, and BI-GRU networks, respectively.

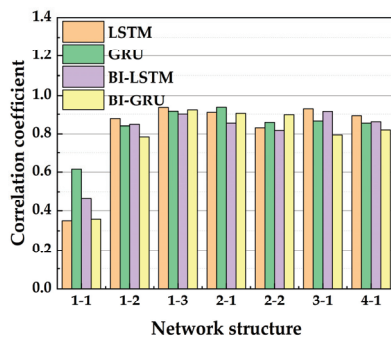


Figure 20. Correlation coefficient (VBM model).

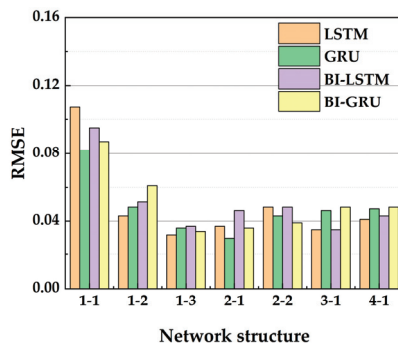


Figure 21. RMSE (VBM model).

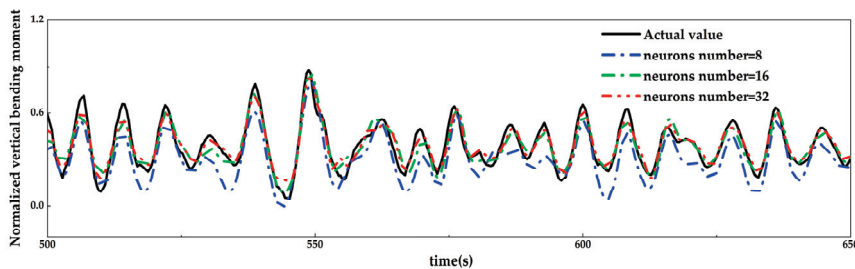


Figure 22. Prediction results under different network structures (LSTM network, layer = 1).

For the HBM prediction model, nine types of network structure were analyzed. A window length of 150 and the ‘adam’ optimizer is chosen. Table 5 illustrates the number of network layers and neurons.

Table 5. Number of network layers and number of neurons (HBM model).

Item	1-1	1-2	1-3	2-1	2-2	2-3	3-1	3-2	3-3
Network Layers	1	1	1	2	2	2	3	3	3
Number of neurons	16	32	64	16-8	32-16	32-32	16-32-16	32-32-32	64-32-16

Figures 23 and 24 illustrate that when the number of layers is set to 1, the GRU network achieves the highest prediction accuracy at $\rho = 0.837$ and $\text{RMSE} = 0.057$. With two layers, the GRU network attains the highest prediction accuracy at $\rho = 0.883$ and $\text{RMSE} = 0.046$. Similarly, with three layers, the GRU network exhibits the highest prediction accuracy at $\rho = 0.901$ and $\text{RMSE} = 0.045$. Figure 25 illustrates the prediction results of the GRU network with varying numbers of hidden layers. The network achieves the highest prediction accuracy when it has three layers and each layer has 32 neurons. The three-layer networks with 32 neurons each are also the optimal network structures for LSTM, BI-LSTM, and BI-GRU networks, respectively.

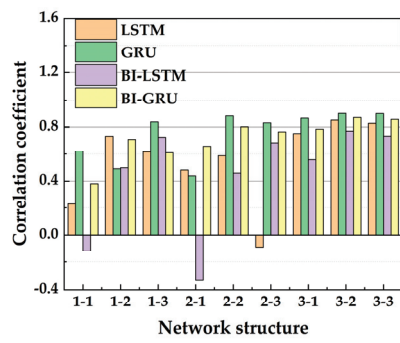


Figure 23. Correlation coefficient (HBM model).

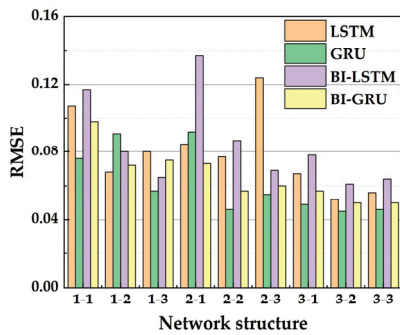
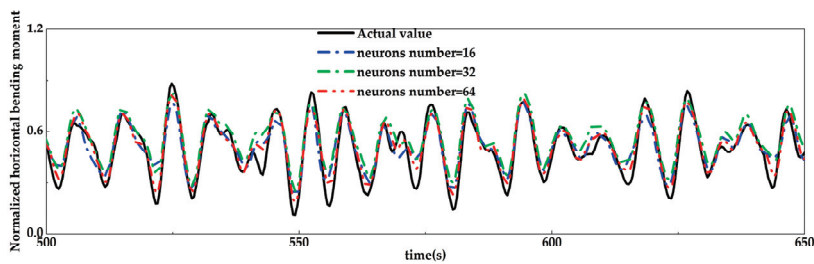
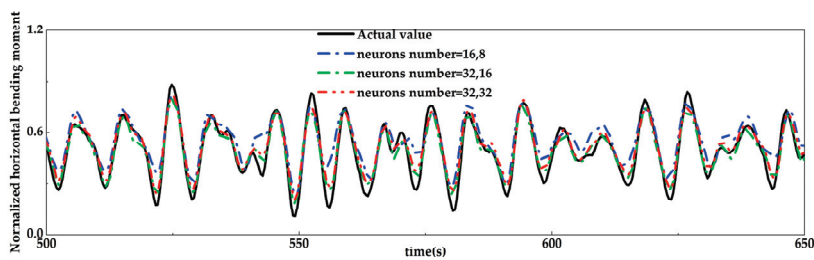


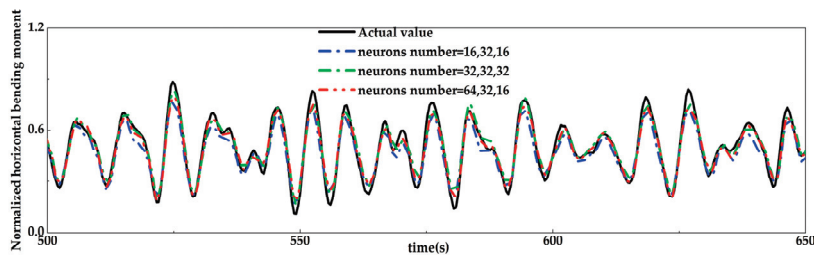
Figure 24. RMSE (HBM model).



(a) Layer = 1



(b) Layer = 2



(c) Layer = 3

Figure 25. Prediction results under different network structures (GRU network).

4.2. Prediction of Different Network Models Using Motion Data with The Same Noise

4.2.1. Numerical Noise

The ship motion data obtained by the hull response monitoring systems (HRMS) is usually accompanied by a large amount of noise. In order to make the simulation data of the testing database similar to the actual ship monitoring data, different levels of numerical noise are artificially added to the simulated data of the testing database. The ship motion data with noise is used as input to verify the prediction accuracy and noise resistance of the ship hull girder load prediction model established in this paper. The motion response to noise can be represented by the following equation:

$$Y = Y^{true} + l \cdot std(Y^{true}) \cdot randn \quad (5)$$

Here, Y^{true} represents the simulation ship motion response, $std(Y^{true})$ represents the standard deviation of the calculated response, l denotes the noise level, and $randn$ refers to a set of random numbers with a mean of 0 and a variance of 1.

Three levels of noise ($l = 5\%$, $l = 10\%$, and $l = 15\%$) were added to the original numerical simulation ship motion data. Figure 26 illustrates a comparison between the time histories of heave motion data and the corresponding data with three levels of noise. It is evident that the deviation from the true value increases with higher noise levels.

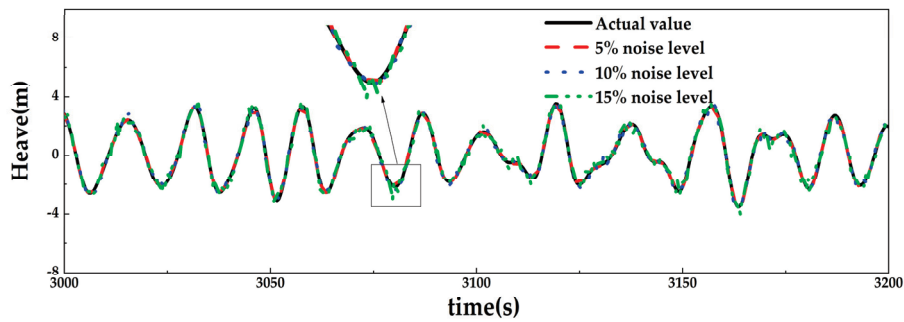


Figure 26. Heave motion data with three levels of noise.

4.2.2. Filtering Technique

When directly using the motion data with added noise for ship hull girder load prediction, the presence of noise often tends to further amplify its impact on the prediction results, leading to greater high-frequency oscillation outputs, as shown in Figure 27. Typically, existing HRMS usually filter real-time monitoring data to mitigate noise effects. The purpose of filtering processing is to eliminate the unrealistic high-frequency oscillation noise. Fast Fourier Transform (FFT) filtering is a noise reduction method widely adopted for experimental data [24,25] and actual monitoring data [26,27]. The basic principle of FFT filtering is to operate on the frequency composition of the signal in the frequency domain to alter its frequency characteristics. By analyzing the frequency, it is possible to filter out the inauthentic high-frequency oscillations that represent noise, thereby extracting pure ship motion data.

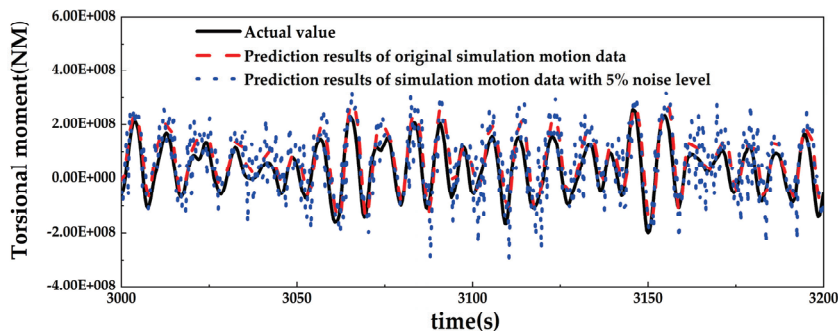


Figure 27. Direct prediction results of 5% noise level.

Spectral analysis is conducted on both the original time-series data and the time-series data with varying levels of noise, as illustrated in Figure 28. It can be observed that selecting a cutoff frequency of 0.2 Hz is appropriate, as it can filter out high-frequency noise at various noise levels. As the noise level increases, the deviation between the filtered data and the original data also increases. To more clearly express the deviation between the filtered data and the original data, the overall standard deviation σ is used to represent the degree of deviation:

$$\sigma = \sqrt{\sum_{i=1}^n (x_i^{\text{original}} - x_i^{\text{filtered}})^2} / n \times 100\% \quad (6)$$

where x_i^{filtered} represents the filtered data at $t = i$, x_i^{original} represents the original data at $t = i$, and n represents the total number of moments. For noise level $l = 5\%$, $\sigma = 4.01\%$. For noise level $l = 10\%$, $\sigma = 7.89\%$. For noise level $l = 15\%$, $\sigma = 12.75\%$.

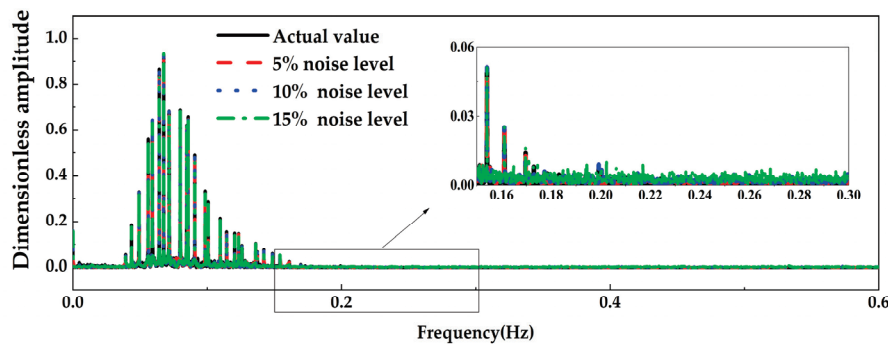


Figure 28. Spectral analysis.

4.2.3. Prediction Results

Through the parameter analysis in Section 4.1, the optimal parameters for the four network models suitable for predicting the hull girder loads have been identified, as shown in Table 6. For the original simulation data without added noise, the prediction results of some models are quite similar. For example, taking the TM prediction as an example, the results from the three-layer GRU network and the single-layer BI-LSTM are both relatively high, with the prediction results not exceeding 1%.

Table 6. Optimal parameter combinations for the four network models.

model	LSTM			GRU		
	window length	optimizer	network structure	window length	optimizer	network structure
TM	150	adam	3, 32-32-32	150	adam	3, 32-32-32
VBM	150	adam	1, 32	150	adam	2, 16-8
HBM	200	adam	3, 32-32-32	150	adam	3, 32-32-32
model	BI-LSTM			BI-GRU		
	window length	optimizer	network structure	window length	optimizer	network structure
TM	100	Nadam	1, 16	150	adam	1, 64
VBM	100	RMSprop	3, 16-32-16	150	adam	1, 32
HBM	200	adam	3, 32-32-32	200	Nadam	3, 32-32-32

In order to further compare the prediction performance of different models on input data with noise, the motion input data with the same noise level are used to validate. The motion data with a noise level $l = 10\%$ are selected. According to Section 4.2.2, the cut-off frequency of 0.2 Hz is chosen to filter the data with noise.

Table 7 illustrates the evaluation index values for the four network models. For the TM prediction model, the GRU network exhibits higher prediction accuracy and noise resistance. For the VBM prediction model, the four network models exhibit similar and relatively high prediction accuracies, the ρ of all four networks remains around 0.87. For the HBM prediction model, compared to the other networks, the GRU network has a slight advantage. Therefore, Table 8 presents the optimal model types and model parameters for predicting torsional moment (TM), vertical bending moment (VBM), and horizontal bending moment (HBM).

Table 7. Evaluation index values for the four network models.

model	LSTM			GRU		
	ρ	RMSE	MAPE	ρ	RMSE	MAPE
TM	0.608	0.089	7.50%	0.660	0.085	7.40%
VBM	0.873	0.050	3.90%	0.875	0.052	4.00%
HBM	0.841	0.052	4.00%	0.886	0.050	3.80%
model	BI-LSTM			BI-GRU		
	ρ	RMSE	MAPE	ρ	RMSE	MAPE
TM	0.583	0.090	8.30%	0.612	0.092	8.50%
VBM	0.876	0.051	4.00%	0.873	0.050	4.00%
HBM	0.838	0.057	4.20%	0.845	0.054	4.30%

Table 8. Optimal network models for hull girder loads prediction.

Model	Type	Window Length	Optimizer	Network Structure
TM	GRU	150	adam	3, 32-32-32
VBM	LSTM	150	adam	1, 32
HBM	GRU	150	adam	3, 32-32-32

4.3. Prediction of the Same Network Models Using Motion Data with Different Noise

In order to test the prediction accuracy of the ship hull girder load models for input data with different noise levels, the original motion response data without noise and with noise level $l = 5\%$, $l = 10\%$, $l = 15\%$ are designed. The evaluation index values are listed in Table 9.

Table 9. Evaluation index values of load case B_MULTI.

Motion Data	TM			VBM			HBM		
	ρ	RMSE	MAPE	ρ	RMSE	MAPE	ρ	RMSE	MAPE
Origin	0.743	0.073	6.20%	0.913	0.042	3.30%	0.945	0.036	2.40%
5% noise	0.729	0.075	6.40%	0.887	0.048	3.60%	0.904	0.046	3.60%
10% noise	0.660	0.085	7.40%	0.873	0.050	3.90%	0.886	0.050	3.80%
15% noise	0.697	0.082	7.10%	0.838	0.062	5.60%	0.850	0.056	4.50%

When the original motion input data is without noise, the test procedure of this paper is similar to the study of Moreira and Guedes Soares [10]. The databases of wave-induced vertical bending moment and shear force prediction model are established based on theoretical methods. When the significant wave height is 3 m, the correlation coefficients of VBM of ADEE ship and Grand America Ro-Ro are 0.8945 and 0.607, respectively. The prediction accuracy of the VBM model of the present study is 0.913. It is obvious that the LSTM model has better prediction performance than the ANN model. As the level of noise increases, the prediction accuracy gradually decreases. For motion input data with a noise of 15%, $\rho = 0.838$ and MAPE = 5.60%. Figure 29 presents the prediction results under

different levels of noise. When the noise level is at 15%, the prediction values closely align with the actual values.

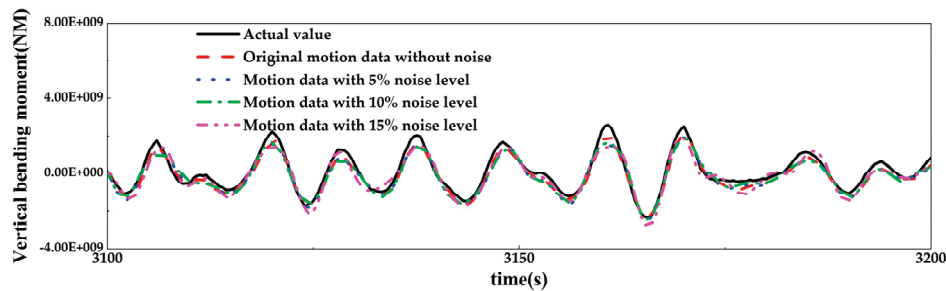


Figure 29. Prediction results under different noise levels (VBM prediction model).

Another advantage of the present paper is its ability to predict horizontal bending moments (HBM) and torsional moments (TM). The prediction accuracy of the HBM model is the highest. For motion input data without added noise, $\rho = 0.945$ and $\text{MAPE} = 2.40\%$. As the level of noise increases, the prediction accuracy gradually decreases. For motion input data with a noise of 15%, $\rho = 0.85$ and $\text{MAPE} = 4.50\%$. Figure 30 presents the prediction results under different levels of noise. When the noise level is at 15%, the prediction values closely align with the actual values.

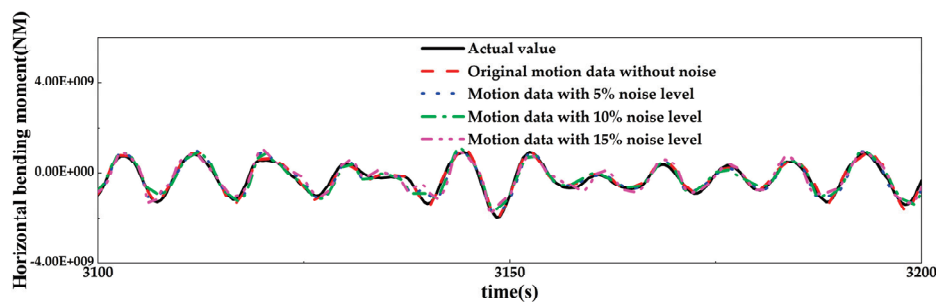


Figure 30. Prediction results under different noise levels (HBM prediction model).

The prediction accuracy of the TM model is the lowest. For motion input data without added noise, $\rho = 0.743$ and $\text{MAPE} = 6.20\%$. As the level of noise increases, the prediction accuracy gradually decreases. For motion input data with a noise of 15%, $\rho = 0.697$ and $\text{MAPE} = 7.10\%$. Figure 31 presents the prediction results under different levels of noise. While the prediction values at a 5% noise level closely align with the actual values, those at 10% and 15% noise levels significantly deviate from actual values.

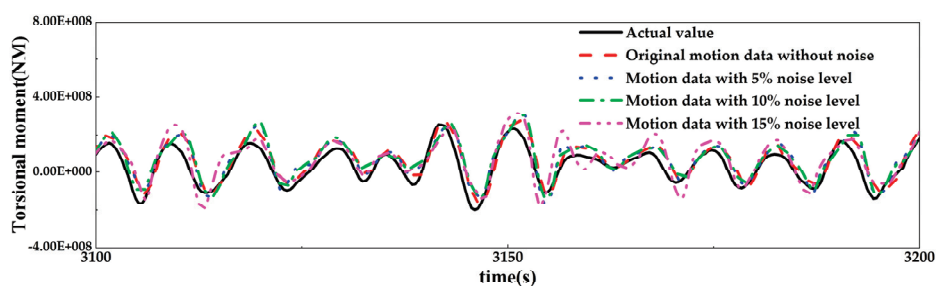


Figure 31. Prediction results under different noise levels (TM prediction model).

In summary, although all noisy data have been subjected to filtering, an increase in the noise level will still lead to poorer prediction results. Filtering techniques cannot filter noisy data to be the same as the original data. Instead, the lower the noise level, the lower the degree of dispersion in the results after filtering, as discussed in Section 4.2.2. When the input data are the original motion response data without noise, the prediction accuracy of

the VBM and HBM models is about 22% higher than the TM model. When the input data are the original motion response data with a 15% noise level, the prediction accuracy of VBM and HBM models remains 20% higher than the TM model.

5. Conclusions

This paper proposes a real-time prediction method of hull girder loads considering the noise in motion monitoring data. Four improved recurrent neural networks are used to establish the hull girder load prediction models. The commercial software Wasim is utilized to acquire databases of ship motion response and hull girder loads in waves. Numerical noise is introduced to the ship's motion response data to emulate real ship motion monitoring data. The following conclusions are drawn:

- (1) The method of controlling variables is used for hyperparameter optimization. Without considering data noise, the optimal network structures of the hull girder load prediction model are respectively obtained based on four improved recurrent neural networks (LSTM, GRU, BI-LSTM, and BI-GRU networks). It found that the most suitable type of recurrent neural network for different load components (TM, VBM, and HBM) is different. The three-layer GRU network with 32 neurons each is the optimal network structure for predicting TM and HBM, and the one-layer LSTM with 32 neurons is the optimal network structure for predicting VBM.
- (2) The real-time prediction model of hull girder load takes ship motion monitoring data as input in practical applications, and motion monitoring data inevitably contain certain noise. It is found that directly using motion monitoring data as input for the real-time prediction of hull girder loads will result in significant prediction errors. Therefore, a numerical filtering method is proposed to preprocess the motion monitoring data. The results indicate that the filtering preprocessing method can significantly improve the prediction accuracy of the model. Using the prediction model that considers the effect of motion data noise, the performance of four improved recurrent neural networks is further analyzed. It is also found that the three-layer GRU network with 32 neurons each is the optimal network structure for predicting TM and HBM, and the one-layer LSTM with 32 neurons is the optimal network structure for predicting VBM.
- (3) The prediction accuracy of the optimal network models for input data with different levels of noise is discussed. It found that although filtering was applied to the original noise data, the prediction accuracy of the model still decreased as the noise level increased. For both the original input data and noisy input data, the prediction accuracy of VBM and HBM is consistently more than 20% higher than that of TM.

Using neural networks to predict hull girder load is a new research direction for the ship structure problem, and it will be of great significance for ship hull intellectualization. There are still some shortcomings in the present study, which need to be improved in the follow-up research.

Due to the lack of actual monitoring data, the model established in this paper lacks validation with actual monitoring data. If actual ship monitoring data can be obtained in the future, the prediction models can be used for further verification.

This paper takes a container ship as an example to propose a prediction method. Other types of ships can also refer to the method proposed in this paper to establish a ship hull girder load prediction model suitable for themselves. The model established in this paper can be further verified for other ship types and other working conditions in future research.

Author Contributions: Conceptualization, Q.W. and L.W.; methodology, Q.W. and L.W.; software, C.L. and Q.W.; validation, X.C. and Q.W.; formal analysis, Q.W.; investigation, L.W. and B.Z.; resources, X.C. and B.Z.; data curation, L.W.; writing—original draft preparation, Q.W.; writing—review and editing, L.W. and X.C.; visualization, C.L. and B.Z.; supervision, X.C.; project administration, L.W. and X.C.; funding acquisition, L.W. All authors have read and agreed to the published version of the manuscript.

Funding: This study is supported by the “National Key Technologies Research & Development Program” (2022YFB3306200); the “National Natural Science Foundation of China” (Grant No. 52171305); National Key Research and Development Program of China [2021YFC2801102]; Joint Fund of Science & Technology Department of Liaoning Province, State Key Laboratory of Robotics [2023-O15]; and Key-Area Research and Development Program of Guangdong Province [2020B1111010004].

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Mohammed, E.A.; Benson, S.D.; Hirdaris, S.E.; Dow, R.S. Design safety margin of a 10,000 TEU container ship through ultimate hull girder load combination analysis. *Mar. Struct.* **2016**, *46*, 78–101. [CrossRef]
2. Amlashi, H.K.K.; Moan, T. A proposal of reliability-based design formats for ultimate hull girder strength checks for bulk carriers under combined global and local loadings. *J. Mar. Sci. Technol.* **2011**, *16*, 51–67. [CrossRef]
3. Almallah, I.; Lavroff, J.; Holloway, D.S.; Shabani, B.; Davis, M.R. Global load determination of high-speed wave-piercing catamarans using finite element method and linear least squares applied to sea trial strain measurements. *J. Mar. Sci. Technol.* **2020**, *25*, 901–913. [CrossRef]
4. Torkildsen, H.E.; Grøvlen, Å.; Skaugen, A.; Wang, G.; Jensen, A.E.; Pran, K.; Sagvolden, G. Development and Applications of Full-Scale Ship Hull Health Monitoring Systems for the Royal Norwegian Navy. In *Recent Developments in Non-Intrusive Measurement Technology for Military Application on Model- and Full-Scale Vehicles*; RTO: Neuilly-sur-Seine, France, 2005.
5. Fanelli, P.; Mercuri, A.; Trupiano, S.; Vivio, F.; Falcucci, G.; Jannelli, E. Live reconstruction of global loads on a powerboat using local strain FBG measurements. *Procedia Struct. Integr.* **2019**, *24*, 949–960. [CrossRef]
6. Zhang, M.; Sun, L.H.; Xie, Y.G. A monitoring method of hull structural bending and torsional moment. *Ocean. Eng.* **2024**, *291*, 116344. [CrossRef]
7. Tayyar, G.T. Overall hull girder nonlinear strength monitoring based on inclinometer sensor data. *Int. J. Nav. Archit. Ocean. Eng.* **2020**, *12*, 902–909. [CrossRef]
8. Kefal, A.; Oterkus, E. Displacement and stress monitoring of a chemical tanker based on inverse finite element method. *Ocean. Eng.* **2016**, *112*, 33–46. [CrossRef]
9. Xu, J.Z. Estimation of wave-induced ship hull bending moment from ship motion measurements. *Mar. Struct.* **2000**, *14*, 593–610. [CrossRef]
10. Moreira, L.; Guedes Soares, C. Neural network model for estimation of hull bending moment and shear force of ships in waves. *Ocean. Eng.* **2020**, *206*, 107347. [CrossRef]
11. Fonseca, N.; Soares, C.G. Experimental investigation of the nonlinear effects on the statistics of vertical motions and loads of a containership in irregular waves. *J. Ship Res.* **2004**, *48*, 148–167. [CrossRef]
12. Xu, W.; Li, H.; Ying, Y.; Zhang, M. Prediction of Total Force and Moment of Ship Based on Improved BP Neural Network. In *Proceedings of the 8th International Technical Conference on Frontiers of Hydraulic and Civil Engineering Technology, HCET 2023*, Wuhan, China, 25–27 September 2023; pp. 1420–1435.
13. Liu, R.X.; Li, H.; Zou, J.; Ong, M.C. Reconstruction and prediction of global whipping responses on a large cruise ship based on LSTM neural networks. *Ocean. Eng.* **2023**, *285*, 115393. [CrossRef]
14. Su, Y.M.; Lin, J.F.; Zhao, D.G.; Guo, C.Y.; Wang, C.; Guo, H. Real-Time Prediction of Large-Scale Ship Model Vertical Acceleration Based on Recurrent Neural Network. *J. Mar. Sci. Eng.* **2020**, *8*, 777. [CrossRef]
15. Li, S.Y.; Wang, T.T.; Li, G.Y.; Skulstad, R.; Zhang, H.X. Short-term ship roll motion prediction using the encoder-decoder Bi-LSTM with teacher forcing. *Ocean. Eng.* **2024**, *295*, 116917. [CrossRef]
16. Fan, G.J.; Yu, P.Y.; Wang, Q.; Dong, Y.K. Short-term motion prediction of a semi-submersible by combining LSTM neural network and different signal decomposition methods. *Ocean. Eng.* **2023**, *267*, 113266. [CrossRef]
17. Liu, Y.C.; Duan, W.Y.; Huang, L.M.; Duan, S.L.; Ma, X.W. The input vector space optimization for LSTM deep learning model in real-time prediction of ship motions. *Ocean. Eng.* **2020**, *213*, 107681. [CrossRef]
18. Xie, Y.J.; Tang, H.S.; Low, Y.M. Deep gated recurrent unit networks for time-domain long-term fatigue analysis of mooring lines considering wave directionality. *Ocean. Eng.* **2023**, *284*, 115244. [CrossRef]
19. Zhang, T.; Zheng, X.Q.; Liu, M.X. Multiscale attention-based LSTM for ship motion prediction. *Ocean. Eng.* **2021**, *230*, 109066. [CrossRef]
20. Liu, H.D.; Liu, Y.; Li, B.; Qi, Z.G. Ship Abnormal Behavior Detection Method Based on Optimized GRU Network. *J. Mar. Sci. Eng.* **2022**, *10*, 249. [CrossRef]
21. Li, H.H.; Xing, W.B.; Jiao, H.; Yang, Z.L.; Li, Y. Deep bi-directional information-empowered ship trajectory prediction for maritime autonomous surface ships. *Transp. Res. Part E-Logist. Transp. Rev.* **2024**, *181*, 103367. [CrossRef]

22. Wang, Z.M.; Qiao, D.S.; Yan, J.; Tang, G.Q.; Li, B.B.; Ning, D.Z. A new approach to predict dynamic mooring tension using LSTM neural network based on responses of floating structure. *Ocean. Eng.* **2022**, *249*, 110905. [CrossRef]
23. Qiao, D.S.; Li, P.; Ma, G.; Qi, X.L.; Yan, J.; Ning, D.Z.; Li, B.B. Realtime prediction of dynamic mooring lines responses with LSTM neural network model. *Ocean. Eng.* **2021**, *219*, 108368. [CrossRef]
24. Xie, H.; Ren, H.; Deng, B.; Tang, H. Experimental drop test investigation into slamming loads on a truncated 3D bow flare model. *Ocean. Eng.* **2018**, *169*, 567–585. [CrossRef]
25. Wang, Q.; Yu, P.; Fan, G.; Li, G. Experimental drop test investigation into cross deck slamming loads on a trimaran. *Ocean. Eng.* **2021**, *240*, 109999. [CrossRef]
26. Lay-Ekuakille, A.; Vendramin, G.; Trotta, A. Spectral analysis of leak detection in a zigzag pipeline: A filter diagonalization method-based algorithm application. *Measurement* **2009**, *42*, 358–367. [CrossRef]
27. Thirumala, K.; Umarikar, A.C.; Jain, T. An improved adaptive filtering approach for power quality analysis of time-varying waveforms. *Measurement* **2019**, *131*, 677–685. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Real-Time Digital Twin of Ship Structure Deformation Field Based on the Inverse Finite Element Method

Pengyu Wei ^{1,2,3}, Chuntong Li ^{1,2,*}, Ze Jiang ³ and Deyu Wang ^{1,2,*}

¹ State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200240, China; wei_pengyu@163.com

² Institute of Marine Equipment, Shanghai Jiao Tong University, Shanghai 200240, China

³ China Ship Scientific Research Center, Wuxi 214082, China; jzcsrc@163.com

* Correspondence: chuntong8856@sjtu.edu.cn (C.L.); dywang@sjtu.edu.cn (D.W.)

Abstract: Digital twins, an innovative technology propelled by data and models, play a seminal role in the digital transformation and intelligent upgrade of ships. This study introduces a digital twin methodology for the real-time monitoring of ship structure deformation fields, based on finite discrete strain data, and a visualization tool framework is developed using virtual reality technology. First, the inverse Finite Element Method (iFEM) is employed to derive the deformation field of the ship structure in real time using sensor strain data. Secondly, the deformation field data obtained based on the iFEM algorithm is converted into general visualization data conducive to interpretation within virtual reality (VR) applications. Lastly, a digital twin software tool is built to enable synchronous responses and interactions between the virtual scene and the physical scene, directly superposing particular virtual objects (data acquired by sensors, computer-aided design (CAD) virtual models, and deformation field cloud images) onto the physical scene in real time. The digital twin tool embodies a virtual reality visualization system framework integrating the physical data measurement, reconstruction, analysis, expression, storage, rendering, and interaction of deformation field data. Through practical application, the flexibility, effectiveness, and compatibility of the developed prototype tool are verified. According to the results, the system can enhance the efficiency of scientific communication, model validation, and interdisciplinary sharing during the analysis and evaluation of the mechanical properties of ship structures.

Keywords: ship structure; inverse finite element method; digital twin; scientific visualization; virtual reality

1. Introduction

In the current era of digitization sweeping across the maritime industry, digital twins have emerged as a crucial component. The domain of marine commerce is considerably impacted by burgeoning globalization tendencies, and in this context, solutions stemming from digital twins emerge as significant pillars of support. These solutions enable ship-building and shipping enterprises to proficiently grapple with sundry challenges, including but not limited to escalating delivery expenses and political compulsions toward decarbonization [1]. These tools serve as significant assets bolstering digitization initiatives. Structural health monitoring (SHM) is a multidisciplinary technical field offering reliable insights into the mechanical behavior of structures under load using measurement data from sensor systems, thus facilitating decision support for structural safety evaluations. Existing stress monitoring systems of hull structures mainly use limited and discrete strain sensors placed directly at monitoring points, which makes it difficult to obtain the full field information of the hull structure and cannot provide a complete stress state description of each detailed structure in the whole ship [2–8].

With the advent of digital twins, a concept that has rapidly risen to prominence, conducting health and safety evaluations rooted in twin models is gradually gaining traction

as a focal point of research. In the recent chronological sphere, initial applications of digital twins within the maritime domain have surfaced [9–12]. The literature evidences the usage of digital twins in the realm of shipping, bifurcating it into two primary classifications [13]. The first category pertains to observation, calibration, and decision support integral to ship operation and modulation. For instance, Coraddu et al. [14] established a numerical model anchored in machine learning to quantify the velocity of damage caused by marine fouling. However, the paradigm necessitated an ample corpus of sample data for the efficient training of the model. The second category is applicable to system integration testing and training. Tofte et al. [15] delineated a simulation system encapsulating virtual humans in the ship pipe installation process, touted for its efficacy in hardware testing and operational training. The utilization of digital twin technology in evaluating structural safety remains in the nascent stages. Bernasconi et al. amalgamated finite element analysis with augmented reality, proffering a twin model dedicated to monitoring fatigue crack evolution in metal composite bonded connections founded on measured strain data [16]. Schirmann et al. [17] instituted a digital twin system for estimating hull structure fatigue, predicated on measured wave response data, serving as an evaluation tool for cumulative impairment endured by the ship.

Leveraging a strain sensor network measurement system to mirror the displacement field of a structure in real time, colloquially known as shape-sensing, constitutes an integral research tactic in SHM. In 2004, LR [18] proffered a scheme for sensor configuration specifically crafted for typical hull structure surveillance in bulk carriers or oil tankers. Classification societies such as the American Bureau of Shipping and Det Norske Veritas have established protocols for sensor types, measurement precision, and data retention requisites for hull monitoring systems [19]. Majewska et al. [20] conferred a methodology for contemporaneous strain/stress monitoring of a sailing ship's foremast employing Fiber Bragg Grating (FBG) sensors. In a notable innovation, Tessler and Spangler [21,22] at NASA Langley Research Centre developed a groundbreaking shape- and stress-sensing algorithm known as iFEM. The utility and precision of the iFEM framework in assessing structural deformation fields have been duly validated [23–26]. Kefal et al. [27] employed the strain data obtained via finite element analysis as input data procured through the finite element analysis as input data within the iFEM algorithm, actualizing deformation surveillance in a typical chemical tanker.

One of the main purposes of employing digital twin models for ship-structure health monitoring is to extract and exploit efficacious data to the maximum possible extent. Commercial software platforms such as CEI Ensign [28] and GLView VTFx [29] intensify the demand for the visual interaction of finite element analysis (FEA) data in virtual reality environments to a certain extent. However, limited interdisciplinary interfaces and non-open-source features limit the application of user innovation scenarios. With the development of VR technology, it is possible to reproduce the real processes of mechanical phenomena. Song et al. developed the middleware CAE2VR [30], a tool adept at transmuting FEA data into virtual visualization files, subsequently crafting a system to disseminate limited heterogeneous FEA data within a virtual reality milieu. Cho et al. [31] proposed a scheme for FEA data conversion and lightweight representation, thereby augmenting the dissemination of FEA outcomes. Taking advantage of the superior memory capacity inherent in supercomputers, Liu et al. [32] transformed the .ODB result file, computed by the Abaqus 6.14 software, into a .vtk visualization file using a Python script, thereby facilitating the parallel visualization of large-scale FEA results in the Paraview commercial software. Through the application of neural networks, Hambli et al. [33] forecasted the real-time deformation of tennis balls and rackets during the collision process, in which users can feel the impact load of the racket through a haptic glove in a virtual reality environment. Although the above-mentioned research has achieved rapid responses to prediction results by training surrogate models, it is still limited by the scale of the finite element model, which may produce unreliable results for untrained inputs. Deciphering ways

to streamline interactions and disseminate FEA data in pragmatic applications continues to pose a formidable challenge [33–36].

Limited by the current level of computer technology, one of the major problems in ship strength digital twin research is how to obtain the global field deformation results of the structure based on limited strain measurement points. Assessing and overseeing the safety status of any structure, such as a ship, typically requires an examination of its inherent mechanical properties. This analysis often involves the development of a finite element model of the target structure, or a reduced-order variant thereof, or the implementation of deep learning techniques. The unwieldy nature of finite element simulations results in the technology falling short of meeting the exigencies for swift response in the monitoring sequence. The adoption of a reduced-order model necessitates a considerable amount of preparation work, demanding repetition when the target structure varies. Deep learning methods require training with an extensive volume of samples, but they may yield unreliable outcomes for untrained inputs. A common feature amongst these three methods is the disjunction between numerical computations and physical scenes.

This paper aims to explore the feasibility of employing limited discrete strain measurement data within physical scenes to effectuate the real-time reconstruction of the structural deformation field, ultimately actualizing the potential of a virtual–real fusion digital twin monitoring system. The self-developed digital twin tool commences with the values gathered from strain gauges on the investigated structures, proceeding with an analysis based upon the iFEM framework. Following the reconstruction of the deformation field, the data are subsequently provided to the VR system to achieve an integration of the monitoring, analysis, and visualization processes. Given that the hull primarily consists of welded thin plate structures, numerical simulations generally utilize quadrilateral or hexahedral meshes to ensure precise computations. Accordingly, our study adopts a four-node inverse-shell element formulation based on Mindlin’s first-order plate theory. Differing from the traditional iterative process of nonlinear numerical simulations, the iFEM method presented herein utilizes real-time measured strain data. This methodology is particularly suited to reconstructing the deformation field during a structure’s linear elastic stage, excluding considerations of material or geometric nonlinearity.

The remaining sections of this paper are organized as follows. In Section 2, the fundamental concepts and overview of the extant digital twin system are explained, and a digital twin framework of virtual–real fusion, a deformation field reconstruction algorithm based on the inverse finite element, and a visualization strategy for the virtual scenes are established. Section 3 introduces the real-time virtual–real interactive platform for ship structure mechanic testing, constructed using the digital twin methodology and validated through comparative verification via physical model test case studies. This section incorporates discussions on various technical issues related to the monitoring of structural mechanical properties, such as the analysis of deformation fields, fusion of physical and virtual data, and virtual visualization of deformation fields. Finally, Section 4 draws the primary conclusions and summarizes the insights gleaned from this study.

2. Methodology

2.1. Real-Time Digital Twin of Ship Structure Deformation Field

Digital twins, utilized for the real-time assessment of ship structure deformation fields, require the following basic elements: a fusion of physical measurements and virtual data, a reconstruction of the deformation field, and a virtual–real fusion visual interactive platform. These components can interlink and bolster a variety of complementary functions, such as data analysis, monitoring, and decision-making. The execution of all analyses, simulations, and visualizations should occur in real time to provide a robust framework for the rapid evaluation of the ship structure’s deformation field. This novel digital twin system harbors the potential to aid operators’ decision-making processes concerning hull structure safety assessments. It focuses on the deterministic identification of the structural deformation state under specified loads, as opposed to statistical values predicated on marine environmental

factors or labor-intensive numerical simulation computations. The system can proffer more granular information on the entire field of the ship structure, thereby mitigating the risk of structural failure induced by the excessive operation of the vessel. As current circumstances dictate, extant digital services are not without their flaws [37]. In the future, the completed system should incorporate assessments of potential large deformations of ship structures under severe sea conditions.

The composition of the digital twin platform for the real-time assessment of ship structure deformation fields is depicted in Figure 1. It encompasses a physical scene and strain data measurement module, an iFEM-based deformation field reconstruction module, a virtual scene construction and data fusion module, a deformation field visualization module, and a digital twin integration platform. The bottom layer of the deformation field reconstruction module integrates the iFEM algorithm to derive global field deformation information about the structure via measured strain data. The virtual sensor measurement module prompts the model within the virtual scene to mimic motion responses consistent with the physical scene, using displacement data derived from the loading device in the physical scene. The deformation field virtual visualization module offers users immersive interactive functionalities within virtual scenes to facilitate the exploration of structural mechanical properties. The digital twin platform amalgamates these diverse functional modules, thereby constructing a digital twin software system for the real-time assessment of ship structural deformation fields through its deployment.

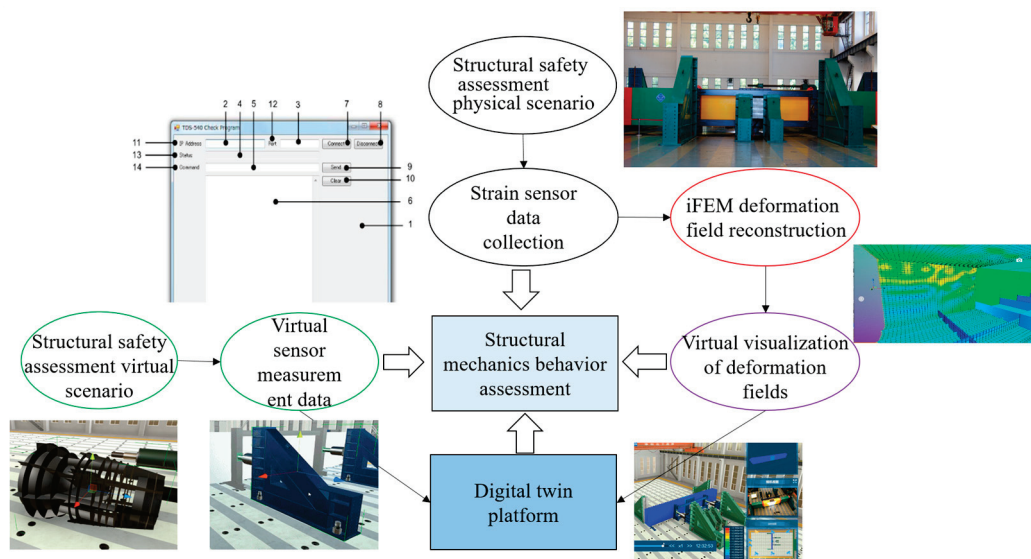


Figure 1. Overall process of real-time digital twin.

2.2. Digital Twin Architecture

Assani et al. [38] described the application of digital twins within the sphere of marine engineering. Upon a perusal of the extant literature, it can be observed that digital twins have varying degrees of integration in imitating physical objects. The first layer is referred to as the digital model, encompassing the digital simulation of the physical entity where the communication between the digital and physical object is wholly manual. The second layer, also known as the digital shadow, ensures the one-way flow of data wherein alterations in the physical model's state are mirrored within the digital copy. The third layer, known as the digital twin, exhibits seamless bi-directional information transmission between the physical and digital objects. The theoretical concept of the digital twin encompasses procuring load data and hull response readings through environmental monitoring to formulate a simulation model. The intersection of virtuality and reality facilitates the conversion of environmental monitoring outcomes into structural stress results, with the

digital model guiding and forecasting for the physical model. As illustrated in Figure 2, the characteristics of digital twins can be described as follows:

- (1) The transition from physical space to virtual space rests on the premise that the digital model possesses high fidelity.
- (2) The implementation approach involves numerical simulation technology and artificial intelligence.
- (3) The key technology lies in facilitating data interaction between physical and virtual spaces.
- (4) The implementation requirement is to meet real-time performance as much as possible.

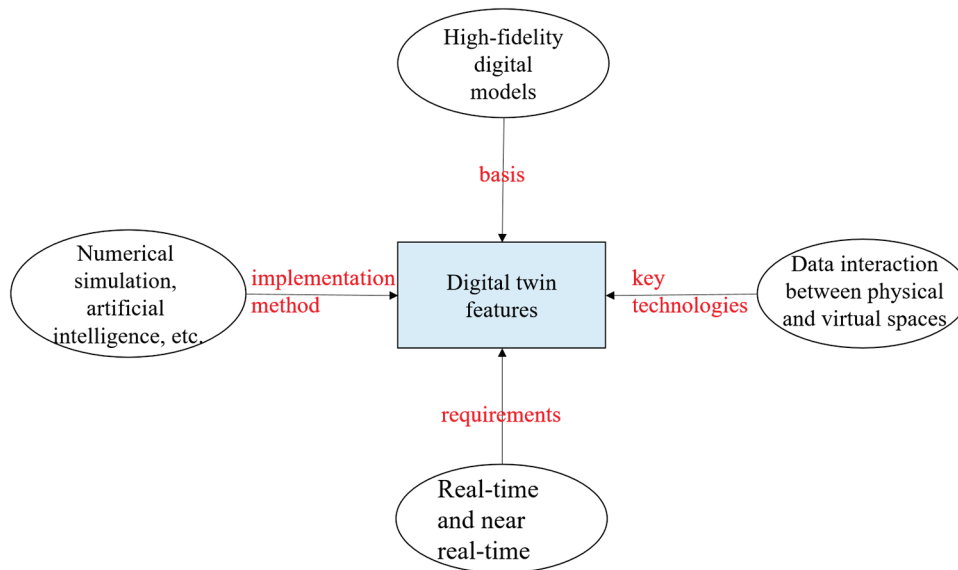


Figure 2. Characteristics of the digital twin approach.

For hull structures, traditional beam theory coupled with finite element and other technologies have offered varying degrees of fidelity in virtual models and numerical simulation results, aligning with the first two characteristics referenced above. The key to realizing the digital twins of ship hull structures currently lies in enhancing the fidelity of virtual models and simulation results, and ensuring rapid responses. Though monitoring the external environment of hull structures is still in its infancy, stress monitoring is relatively mature. Consequently, strain measurements can serve as the data source for the digital twin system dedicated to ship structure safety monitoring. Given the impracticality of installing strain sensors at all points in the ship structure, particularly within narrow and enclosed spaces, utilizing limited strain measurement locations to procure the global field information of the structure poses a significant technical challenge. Additionally, the finer the numerical model, the higher the computational cost, which contradicts the real-time nature of digital twins. Hence, resolving the swift response within the digital twin platform remains an important issue. In this research, the conceptual implementation of the digital twin for a ship hull structure safety assessment (Figure 3) envisages using partial stress monitoring points to effectuate the deformation field reconstruction, supplemented with displacement monitoring data in the physical environment. Such data are incorporated into the construction and simulation technology of the digital hull structure model, and the structural state of the ship's global field is obtained through an efficient computation of the deformation field reconstruction. In the meanwhile, relevant measured data and reconstruction results are relayed back to engineering and experimental designers via virtual reality technology.

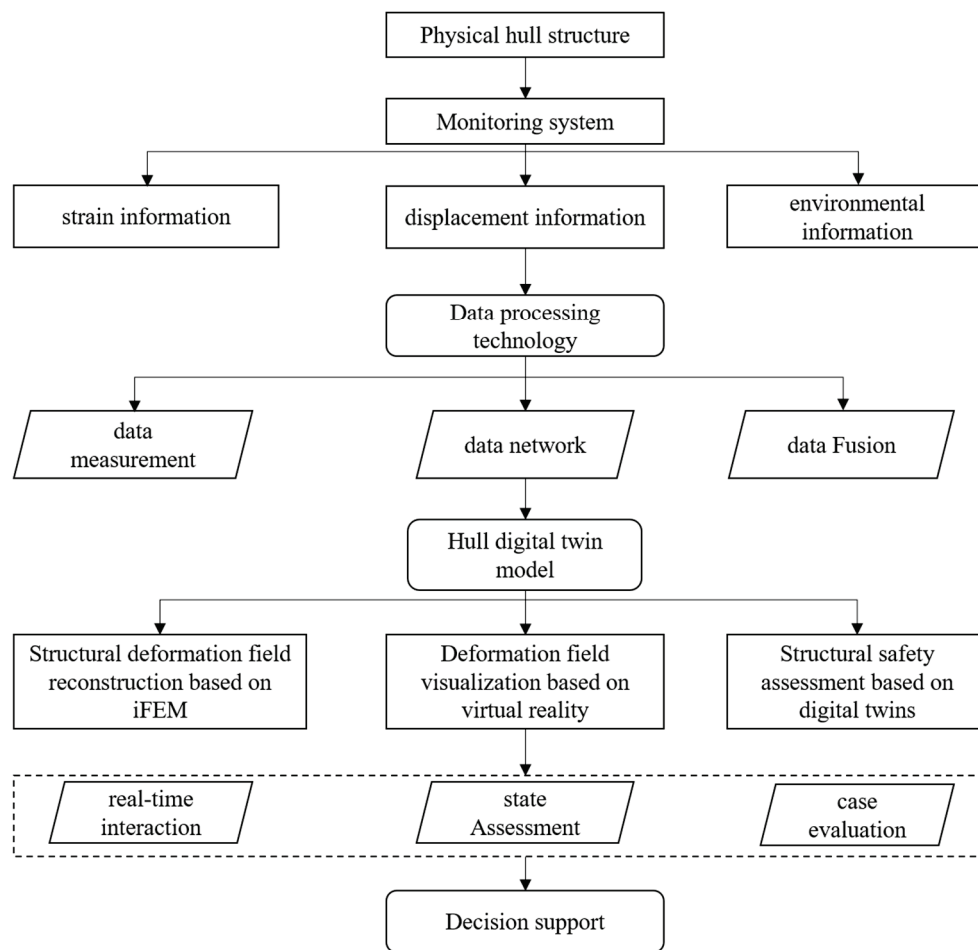


Figure 3. Real-time digital twin technology flow chart for hull structure safety assessment.

Digital twin epitomizes the cutting-edge evolution in implementing digital modeling, simulation, and industrial internet of things (IoT) technology, encompassing all digital models of the physical system to simulate, predict, and assess key activities interrelated with physical measurement data. The content of the digital twin is organized here as the embodiment of the physical entity, interconnectedness of states and data harvested from the entity, and simulation or scrutiny of behavior. A digital twin includes all entities and virtual models shared in the simulation. Sensor measurement data collected from the system serve in the analysis and simulation of the structural mechanical behavior. The digital twin developed in this research facilitates a bidirectional flow of data information between the actual hull structure and its virtual model. Additionally, for a digital twin platform of high-fidelity simulation models across multiple domains, it is prudent to adopt a phased progression in development. Functional modules are independently developed and then associated with the digital twin platform to strike the necessary equilibrium between fidelity and simplification.

2.3. Inverse Finite Element Formulation for Shells

The real-time, rapid, and accurate acquisition of structural state information is crucial for early warning mechanisms and bolstering ship safety. Most SHM methods do not take into account the structure's topology and boundary conditions. The iFEM reconstructs the overall displacement field of the structure in real time from strain data gathered from a network of sensors stationed at various points on the structure. Unlike other developed SHM systems, the iFEM method possesses broad applicability to intricate structures influenced by complex boundary conditions in real time [22]. Given that the strain measurements represent the sole input in the iFEM formulation and the constructed strain-displacement

relationship does not involve the structure's material properties and load information, there is no requirement for prior knowledge of material characteristics and load information, making it well-suited for complex structures. The four-node quadrilateral inverse-shell (iQS4) element formulation is established under the assumption of Mindlin's first-order plate theory. Considering the practical application in marine structures, the measured strain data in this study serve as the input for the iFEM algorithm, with the structure's deformation field as the output data.

2.3.1. Quadrilateral Inverse-Shell Element

This section employs the iQS4 element to demonstrate the derivation process of the iFEM formula. As depicted in Figure 4a, the iQS4 element, global coordinate system XYZ, and local coordinate system xyz are illustrated. The origin of the xyz system is located at the centroid of the element's middle surface quadrilateral. Nestled at the centroid of the middle surface quadrilateral of the element and furnished with six degrees-of-freedom (DOF) per node, as highlighted in Figure 4b, the iQS4 element encompasses translations u_i, v_i, w_i in addition to rotations $\theta_{xi}, \theta_{yi}, \theta_{zi}$. The theoretical strain of the structural element is divided into plane strain $e(u^e)$, bending curvature $k(u^e)$, and transverse shear strain $g(u^e)$, where u^e symbolizes the displacement exhibited by the four nodes. The formulation of the iQS4 element is actualized initially by composing the membrane and bending curvature matrices separately, ultimately amalgamating them for exportation. The relevant calculations of the iQS4 are procured relying on Mindlin's plate theory, considering the influence of shear deformation. According to the assumptions of Mindlin's first-order plate theory, the relevant derivatives of the three components of the displacement vector are calculated. Subsequently, the strain-displacement relationship of linear elastic theory is deduced as follows:

$$\begin{aligned} e(u^e) + zk(u^e) &= B^m u^e + zB^k u^e = (\varepsilon_{xx} \varepsilon_{yy} \varepsilon_{zz})^T \\ g(u^e) &= B^s u^e = (\gamma_{xz} \gamma_{yz})^T \end{aligned} \quad (1)$$

where $u^e = [u_1^e \ u_2^e \ u_3^e \ u_4^e]$, $u_i^e = [u_i \ v_i \ w_i \ \theta_{xi} \ \theta_{yi} \ \theta_{zi}]^T$ with $(i = 1, 2, 3, 4)$, and B^m is a matrix that stands for the derivatives of the shape functions associated with the membrane behavior. Conversely, B^k, B^s matrices manifest the corresponding derivatives of shape functions designed to delineate the bending behavior of the element.

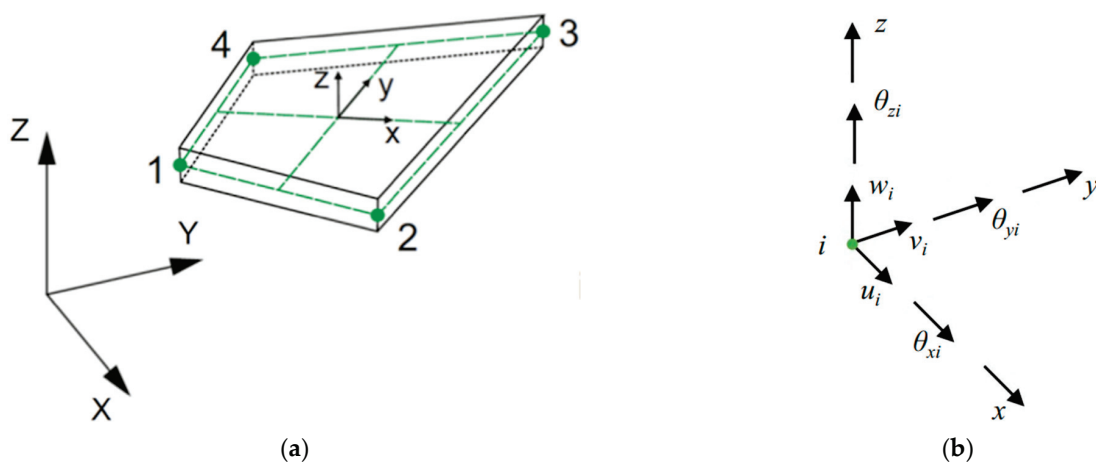


Figure 4. Four-node inverse-shell element. (a) iQS4 element showing global and local coordinate systems; (b) Nodal DOFs in the local coordinate system xyz.

2.3.2. Input Data from In Situ Strain Sensors

The input for the iFEM is the measured strains at the center points on the top and bottom surfaces of each inverse element, as illustrated in Figure 5. Commonly used strain

sensors incorporate strain rosettes or FBG sensor networks. The distinct attributes of an FBG system include its lightweight nature, swift operation, immunity to electromagnetic interference, and a lack of necessity for recalibration post installation. Tessler and Spangler (2005) pointed out that reference plane strains e_i^ε and curvatures k_i^ε could be calculated utilizing measured surface strains [39]. In Equations (2) and (3), ε_{xx}^+ , ε_{yy}^+ , γ_{xy}^+ denotes the measured strain on the upper surface of the element, ε_{xx}^- , ε_{yy}^- , γ_{xy}^- represent the measured strain on the lower surface of the element, and h is the distance from the middle surface to the surface. Transverse shear strain g_i^ε cannot be measured directly through strain rosette sensors. However, in the context of plate and shell structures, the value of transverse shear strain is much smaller those of plane strain and bending curvature, thereby rendering the influence of transverse shear strain negligible in the iFEM formulation.

$$e_i^\varepsilon = \frac{1}{2} \left(\begin{pmatrix} \varepsilon_{xx}^+ \\ \varepsilon_{yy}^+ \\ \gamma_{xy}^+ \end{pmatrix} + \begin{pmatrix} \varepsilon_{xx}^- \\ \varepsilon_{yy}^- \\ \gamma_{xy}^- \end{pmatrix} \right)_i \quad \text{with } (i = 1, \dots, n) \quad (2)$$

$$k_i^\varepsilon = \frac{1}{2h} \left(\begin{pmatrix} \varepsilon_{xx}^+ \\ \varepsilon_{yy}^+ \\ \gamma_{xy}^+ \end{pmatrix} - \begin{pmatrix} \varepsilon_{xx}^- \\ \varepsilon_{yy}^- \\ \gamma_{xy}^- \end{pmatrix} \right)_i \quad \text{with } (i = 1, \dots, n) \quad (3)$$

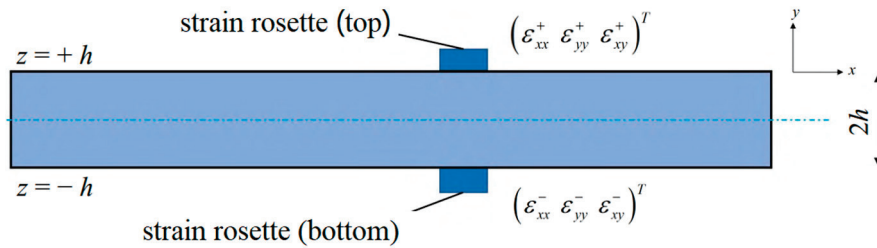


Figure 5. Strain rosettes instrumented on the top and bottom of iQS4 elements.

2.3.3. Weighted Least-Squares Function

At the core of the iFEM lies the construction of a least-squares function $\phi_e(u^e)$ between the measured strains of an element and the theoretical strains. By considering the membrane, bending, and transverse shear deformation of the element, the least-squares function is minimized with respect to the nodal displacements using the variational principle. Subsequently, an estimated value for the node displacement is procured and deemed reflective of the measured displacement.

$$\phi_e(u^e) = w_e \|e(u^e) - e^\varepsilon\|^2 + w_k \|k(u^e) - k^\varepsilon\|^2 + w_g \|g(u^e) - g^\varepsilon\|^2 \quad (4)$$

In Equation (4), w_e , w_k , w_g are weight coefficients. The correct use of the weighting constant is especially critical for problems involving a relatively small number of strain measurement points. Setting the weighting constant at 1 is recommended when the element possesses a corresponding strain measurement value. The square norm depicted in Equation (4) can be rephrased in the context of a normalized Euclidean norm as follows:

$$\begin{aligned} \|e(u^e) - e^\varepsilon\|^2 &= \frac{1}{n} \iint_{A^e} \sum_{i=1}^n (e(u^e)_i - e_i^\varepsilon)^2 dx dy \\ \|k(u^e) - k^\varepsilon\|^2 &= \frac{(2t)^2}{n} \iint_{A^e} \sum_{i=1}^n (k(u^e)_i - k_i^\varepsilon)^2 dx dy \\ \|g(u^e) - g^\varepsilon\|^2 &= \frac{1}{n} \iint_{A^e} \sum_{i=1}^n (g(u^e)_i - g_i^\varepsilon)^2 dx dy \end{aligned} \quad (5)$$

In the formula, A^e is the area of the element. For inverse elements without strain sensors, the weighting constant is set to 10^{-5} . These weighting constants guarantee satisfactory regularization of Equation (4), even under circumstances where the iFEM model presents scant measurement strain data. This allows for necessary interpolation linkages between inverse-shell elements equipped with strain rosettes. Nonetheless, augmenting the quantity of elements for strain-free measurements has the potential to impair the precision of the deformation field reconstruction. With these assumptions, the least-squares function is minimized using the variational principle, i.e., through the extraction of partial derivatives relative to nodal displacements:

$$\frac{\partial \phi_e(u^e)}{\partial u^e} = k^e u^e - f^e = 0 \quad (6)$$

where k^e , f^e are the inverse stiffness matrix and load column matrix of the element, respectively. The overall displacement of the structure can be calculated from both the overall inverse stiffness matrix and the corresponding inverse load column matrix. Once the local matrix equations are defined, the element contributions to the global linear equation system of the discretized structure can be obtained as follows:

$$KU = F \quad (7)$$

where K is correlated with the measured strain value, U embodies the global nodal displacement vector, and F represents a global function of the measured strain value.

2.4. Visualization and Visual Interaction

The use of VR technology in the evaluation of engineering analysis results has attracted increasing attention [40]. Some existing shared formats can indeed enhance the exchangeability of deformation field data in VR scenarios. However, they are still relatively inefficient. Therefore, the concept of a new practical shareable neutral data architecture is proposed for the effective sharing of multidisciplinary data in collaborative environments. The most critical requirements are summarized below:

- (1) Improve access to data sources and automation level.
- (2) Provide higher data processing efficiency and facilitate verification by design and engineering personnel.
- (3) Enhance the immersive exploration of data results.
- (4) Enhance the interactivity of simulation results.
- (5) Traceability and consistency of data result conversion.

This study attempts to propose a VR-based visualization postprocessing system for the virtual visualization of strain field data, thereby enhancing the attention of engineering designers and managers to the structural change results. Among them, the developed workflows range from reading raw data to displaying visualizations in a virtual environment (VE). Hence, a virtual visualization framework was established, as shown in Figure 6. The virtual visualization system consists of three modules, which are a discrete mesh model reconstruction module, deformation field data processing module, and virtual interaction module. Through customized data interfaces between different modules, data interaction and management during system integration can be realized. The discrete mesh model reconstruction module primarily completes the reconstruction of the target structure in the virtual scene, laying the foundation for the visualization of the deformation field cloud diagram. The deformation field data processing module mainly completes readings and node assignments for the deformation field data obtained by the iFEM algorithm. The virtual interaction module renders and interacts with the deformation field and observes and evaluates the structure through various interactive methods such as segmentation, rotation, slicing, and extraction.

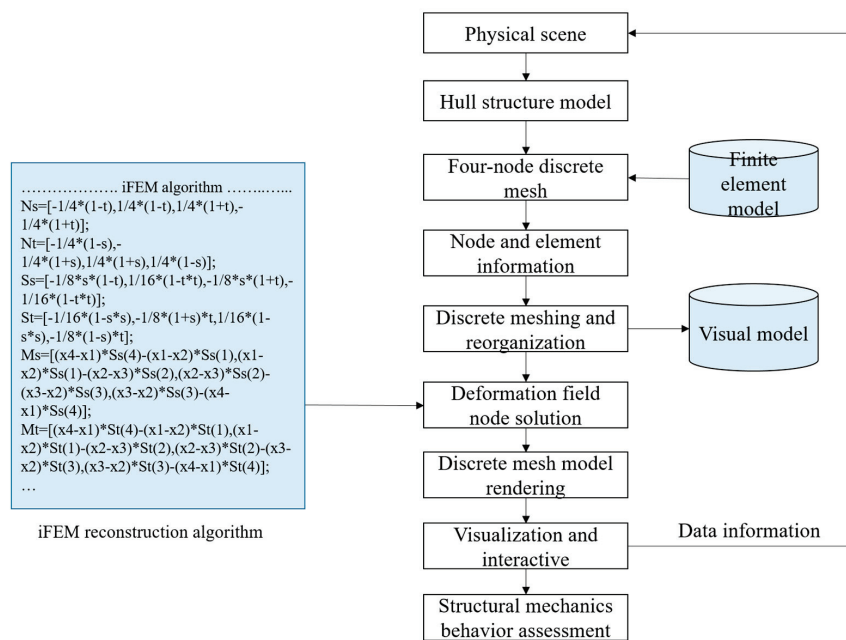


Figure 6. Deformation field virtual visualization implementation process.

The process of visualizing the discrete mesh model and deformation field data involves nodes, elements, vertex sequences, normal sequences, edge sequences, and texture sequences. Shareable models must minimize files while retaining necessary information. Since the Unity 3D platform does not support quad-mesh rendering, the meshing and reorganization algorithms are also embedded. Delaunay triangulation (DT) is the most common unstructured triangulation method in many applications [41], which maximizes the minimum angle of any triangle [42]. For large-scale models, the subdivision and reorganization algorithm was designed in parallel, and Python's multi-processing module is used to call multiple processes to achieve full utilization of system resources. In the division, elements are evenly distributed among different processes, and their array intervals are calculated. During the reorganization process, data blocks are evenly distributed to different processes for sub-model reorganization, where each process reads the shared data without writing.

3. Case Studies for Real-Time Digital Twin

3.1. Application Objects and Test Preparation

This paper targeted a stiffened plate structure of an actual hull, aiming to validate the reconstruction algorithm, deformation field visualization, and digital twin platform. The employed material was Q235 steel. The stiffened plate structure contained two types of stiffeners: T-shaped bar and flat steel, as represented by a geometric schematic in Figure 7. The main geometric parameters of the stiffened plate structure test object are depicted in Tables 1 and 2.

Table 1. Main scale parameters of stiffened plate model (unit mm).

Length	Width	Panel Length	Panel Width
400	416.7	100	69.4

Table 2. Stiffened plate model structural parameters (unit mm).

Plate Thickness	Number of T Profiles	Number of Flat Bars	T Profile Parameters $h_w \times t_w/b_f \times t_f$	Flat Steel Parameters $h_w \times t_w$
6	1	4	$22 \times 6/17 \times 6$	10×6

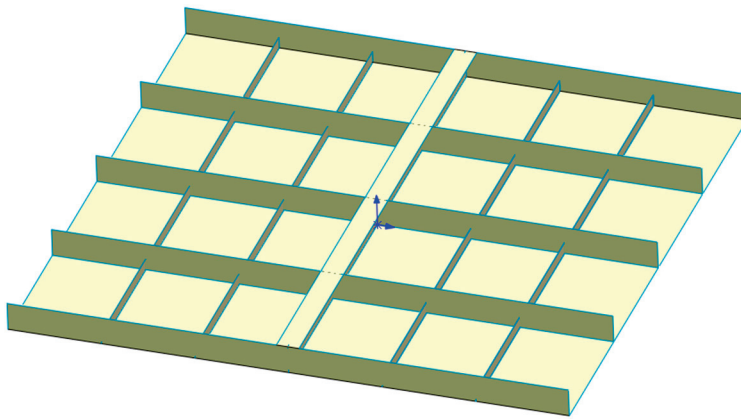


Figure 7. Schematic of 3D model of stiffened plate.

Figure 8 illustrates the configuration of the strain rosettes and displacement sensors, with displacement meters (w1–w6) placed in areas of significant vertical displacement. Strain gauges were located in areas of high stress levels, incorporating triaxial strain rosettes (T1–T24) at the center of the plate grid, and uniaxial strain gauges (S1–S8) at the mid-span of the reinforced rib and T-profile web. Figure 9 showcases the stiffened plate structure test scheme, primarily comprising longitudinal reaction frames, crossbeams, and hydraulic cylinders. The reaction frame was fixed to the ground, while the hydraulic cylinder transferred the load to the crossbeam, thereby applying axial compression load onto the stiffened plate. Displacement sensors measure the stroke of the hydraulic cylinder. The surface load imposed on the stiffened plate structure was facilitated through the use of sandbags.

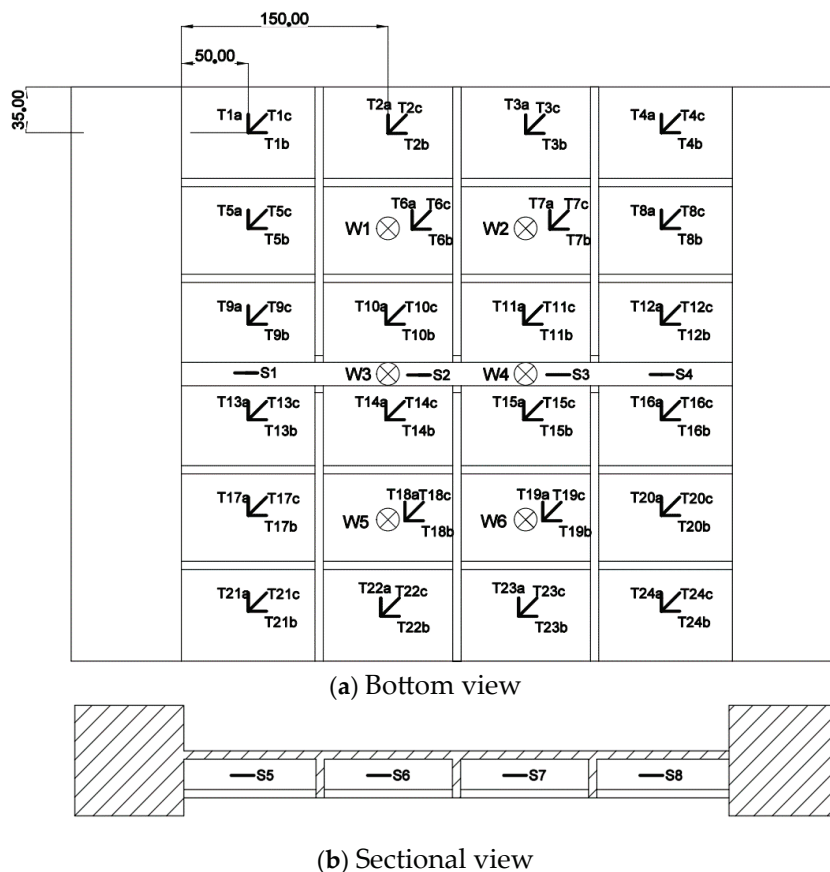


Figure 8. Stiffened plate model strain gauge and displacement gauge layout diagram.

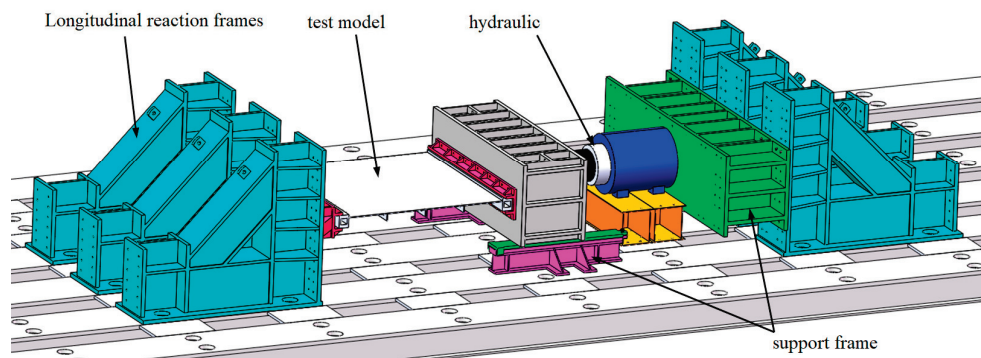


Figure 9. Loading scheme of the test model.

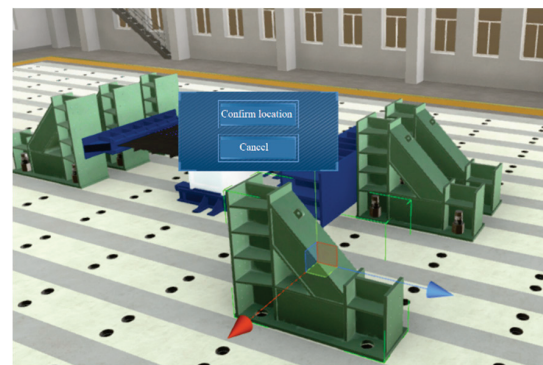
3.2. Ship Structural Mechanics Test Digital Twin

Considering the increasing importance of digital twins within the ship industry, a digital twin data integration method adaptable to different data classifications is proposed, supporting data exchange and interoperability between various functional modules. Within the context of content, a framework was established for both physical scene representation and status alongside a methodology for simulating the mechanical behavior of the target structure. Digital twin data are arranged into independent databases, accompanied by textual descriptions outlining their purpose and attributes. Through this modular approach, the independence of distinct data resources is preserved while enhancing the data exchange. The digital twin framework, developed based on these principles, was applied to a case study examining a hull structural mechanics tests, aiming to serve as a starting point for more complex future applications. Digital twin data were associated with deformation field reconstruction algorithms and virtual visualization algorithms, developed within a proprietary platform. These were employed for monitoring and controlling the mechanical tests of the target structure model, thus offering a superior degree of service maturity and a wider spectrum of data information in comparison to mere online monitoring tools.

In accordance with the categories expounded upon in Section 2.2, digital twin content was divided into four groups: physical models, sensor monitoring, virtual scenes, and behavior models. Data patterns were discerned and mapped using referencing tags retained in an array of databases. Physical models and virtual scenes were linked and mapped through 3D models, sensors, location coordinates, and identification tags in the hull structural mechanics test scene, as shown in Figure 10. The 3D models in the virtual scene exhibited comprehensive detailing (e.g., side and bottom), while the overall motion response was monitored through the assimilation of collision and displacement data within the physical scene. The behavior model was mainly realized through deformation field models, constructed by the iFEM algorithm. An overview of the digital twin pattern is presented in Figure 11.

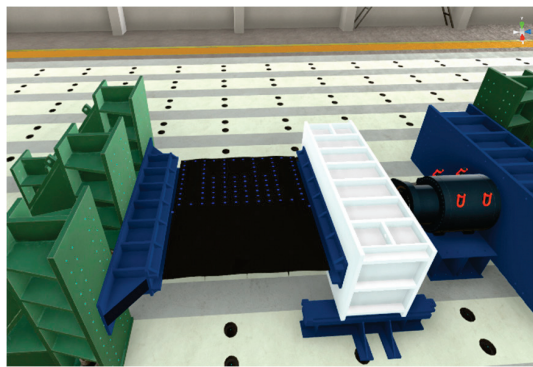


(a) Physical test scene

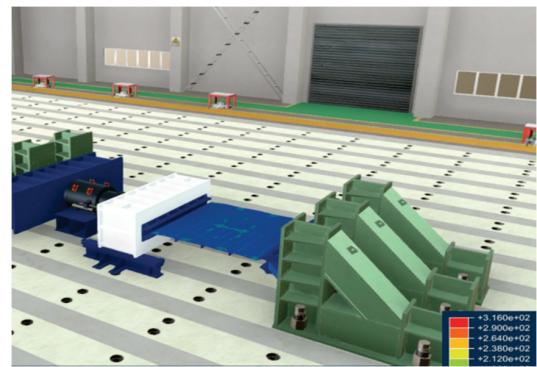


(b) Virtual assembly process

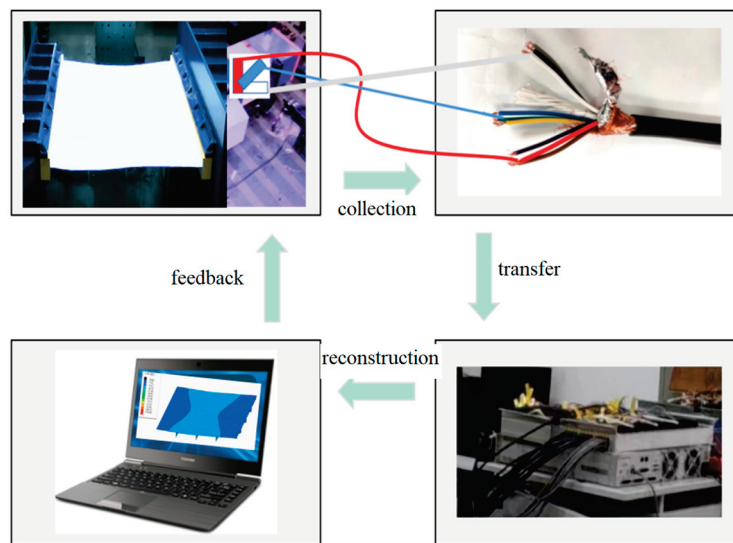
Figure 10. Cont.



(c) Virtual sensor arrangement



(d) Virtual test scene

Figure 10. The construction process of digital twin scenarios.**Figure 11.** Data transfer of digital twin models.

3.3. Visualization and Interaction

The virtual visualization system architecture adopted a multi-module design method and independent interfaces for data transmission between modules such as mesh segmentation and reorganization, data visualization, visualization and interaction functions, which not only ensured the relative independence and functional scalability of each module, but also standardized data communication between systems. This study adopted the divide-and-conquer approach [43] to reconstruct the finite element mesh in the virtual environment. Figure 12 shows the model reconstruction and rendering process based on point arrays in the virtual environment. The node index, node coordinates, element index and node connection order of the mesh were extracted, and the node array was drawn in the virtual visualization space. Through the mesh index, the quadrilateral elements were spatially divided to construct subsets, a Delaunay triangle subgrid was constructed in each subset, and the subgrids were merged recursively. The mesh reconstruction prioritizes the original triangular mesh and construction order in the finite element data to avoid local shape distortion. By mapping the numerical solutions of physical quantities on nodes into the RGB color space [44], a color cloud image for the deformation data visualization was constructed. The virtual visualization system was built and released based on Unity 3D 2019 software and was developed using C# and Python, which run on the Microsoft Windows operating system. This study used a surround-screen virtual test environment provided by the Virtual Reality Laboratory of Ship Digital Design of Shanghai Jiao Tong University in China.

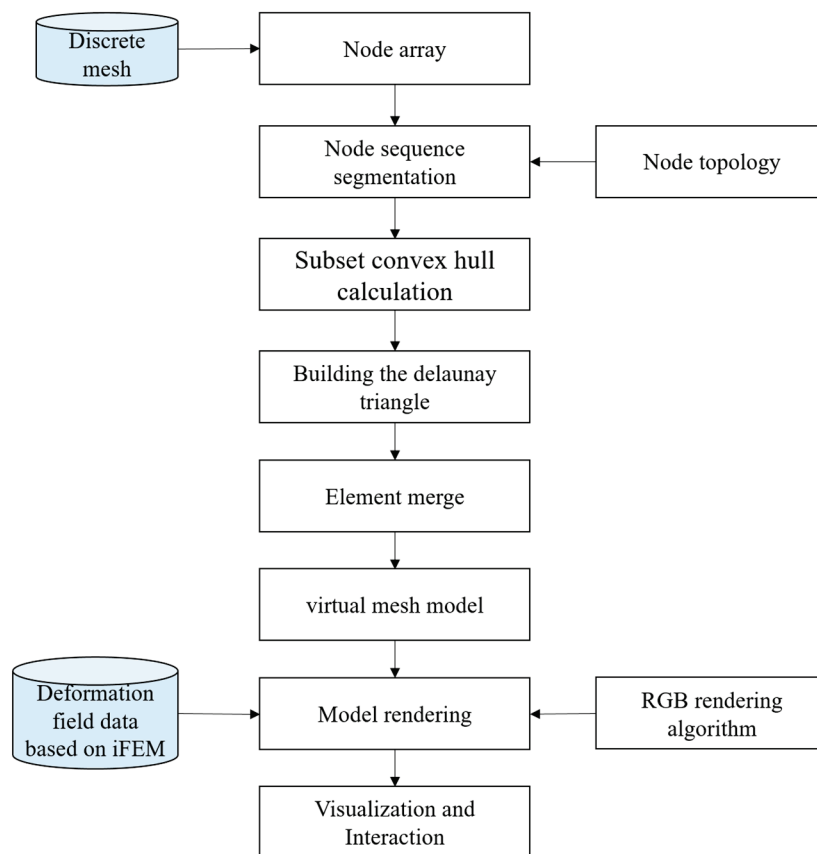


Figure 12. Model reconstruction and rendering based on point arrays.

Overall, in the visualization strategy of the virtual visualization system, the mesh nodes were loaded first. Then, the finite element mesh was reconstructed according to the connection sequence of each element. Finally, the node solutions and element solutions were extracted from the deformation filed database for assignment to realize the model rendering and visualization. It should be emphasized that the rendering of the model was interpolated through the color of the triangle patch nodes, and finally, the rendering of the entire finite element model was achieved. For the visual interaction module, the aim was to facilitate the exchange of data between humans and machines. The operation methods included model scaling, perspective slicing, rotation, cropping, and mesh hiding, which can be carried out by mouse, keyboard, or VR gamepad. As for the deformation file results, they can be checked using a handheld cutter or clipped using a virtual cube. The physical model test application scenario is shown in Figure 13.

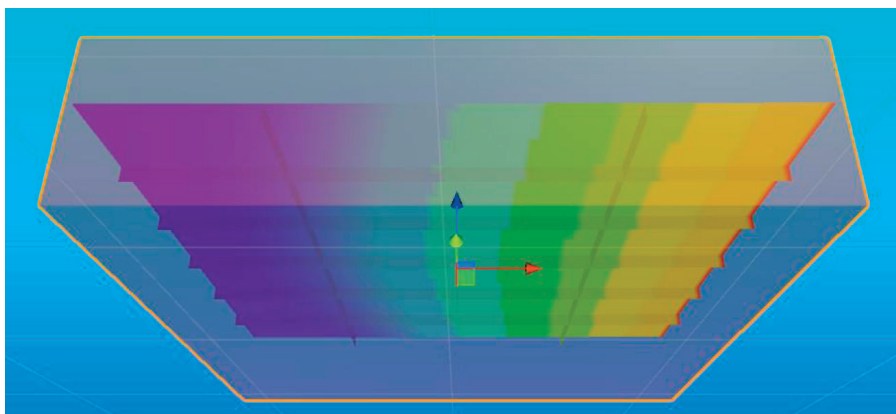


Figure 13. Deformation field cloud diagram.

3.4. Real-Time Digital Twin Platform

Based on the Unity3D platform, a graphical interface for the real-time digital twin of ship structural mechanics is provided to users, thereby enabling remote data access and control over physical tests. The interface displays four elements: the central 3D scene, the sensor data information window on the left, the 3D deformation field visualization graphic window, the physical scene video window on the right, and the control panel at the top (Figure 14). The 3D visualization is nearly synchronous with the physical scene test process, exhibiting real-time displays and facilitating six DOFs in rotation and scaling operations. The most basic function of the digital twin involves the supervision and control of the experiment. The interactive interface provides two monitoring methods: data and video. Cameras are placed to allow users to remotely view the physical test scene. The displacement data of the loading device provides users with the most direct load information, frequently employed to authenticate the functionality of the propulsion system and the loading state of the model, and to corroborate the synchronicity of the virtual assembly model with the physical model during the test. For monitoring structural mechanical behaviors, real-time deformation reconstruction results and comparisons with measured displacement data are provided through the iFEM algorithm. The left side of Figure 14 shows output strain measurement point data, a deformation reconstruction, and a comparison with measured data. This feature proves particularly useful during experimentation, in facilitating debugging, and during testing of the experimental setup.



Figure 14. Real-time digital twin platform for ship structure mechanic testing.

4. Discussion

Drawing on our experience, promoting data exchanges and interoperability has facilitated the construction of the synchronized motion response of the virtual assembly scene relative to the physical testing process. Ensuring the effective communication of assumptions among test engineers, whether through detailed software documentation or verbal directives, enables operators to respond adeptly and diminishes the likelihood of test failures. A significant theme for subsequent research involves exploring methods for executing more complex application scenarios within the digital twin framework, with an ultimate aim to expand its applicability to a broader digital architecture. However, it is worth noting that our current research methodology lacks the capability to solve the reconstruction problem associated with nonlinear large deformations of the structure. Currently, there are two potential solutions to address this issue. The first solution involves using a method similar to Newton's iteration, which takes into account the nonlinear problems arising from geometry and materials. However, this method may result in extended response times during the reconstruction process. The second solution proposes correcting

the reconstruction results using a fast-order reduction calculation method. This approach effectively ensures a rapid response during the reconstruction process.

5. Conclusions

Powerful online monitoring technology serves as a pivotal catalyst for remote control centers, particularly when integrated with dynamic perception algorithms. Practical case studies demonstrate the potential of digital twins in augmenting the monitoring of ship structural mechanic characteristics. The utilization of immersive visualization simplifies the conveyance of the mechanical behavior and state of a structure, superseding traditional methods such as the exhibition of logs and diagrams or the triggering of alarms. The responsive interaction employed in this paper was actualized through a high-frequency data exchange communication system interfacing with the physical setup. A modular methodology was employed to link the deformation field monitoring system to the digital twin. The iFEM approach was used to acquire real-time, global field deformation information of the target structure, eschewing reliance solely on independent data from key discrete measurement points. On one hand, the integration of the iFEM algorithm facilitates rapid responses to strain data, offering more flexibility compared to traditional time-consuming numerical simulations and enhancing the capabilities of the digital twin as a unified integrated system. On the other hand, the proposed immersive virtual visualization interaction scheme revolutionizes the exploration of structural mechanical behaviors and enhances the workflow of virtual reality-supported decision making. Unforeseen interactions may surface during the operation of distributed subsystems.

Author Contributions: Conceptualization, C.L.; Methodology, C.L. and D.W.; Software, P.W. and Z.J.; Validation, C.L. and D.W.; Investigation, Z.J.; Data curation, P.W. and Z.J.; Writing—original draft, P.W.; Writing—review & editing, C.L. and D.W.; Visualization, Z.J.; Supervision, Z.J. All authors have read and agreed to the published version of the manuscript.

Funding: This paper was supported by the project of National Natural Science Foundation of China (Grant No. U2241266), Innovative Research Foundation of Ship General Performance (26822212), Fundamental Research Funds for the Central Universities, and The National Key Research and Development Program of China (2022YFB3404800).

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Fonseca, Í.A.; Gaspar, H.M. Challenges when creating a cohesive digital twin ship: A data modelling perspective. *Ship Technol. Res.* **2021**, *68*, 70–83. [CrossRef]
2. Momdoro, A.; Soliman, M.; Frangopol, D.M. Prediction of structural response of naval vessels based on available structural health monitoring data. *Ocean. Eng.* **2016**, *125*, 295–307. [CrossRef]
3. Johnson, N.R.; Lynch, J.P.; Collette, M.D. Response and fatigue assessment of high speed aluminium hulls using short-term wireless hull monitoring. *Struct. Infrastruct. Eng.* **2018**, *14*, 634–651. [CrossRef]
4. Jo, J.; Jo, B.; Khan, R.M.A.; Kim, J. A cloud computing-based damage prevention system for marine structures during berthing. *Ocean. Eng.* **2019**, *180*, 23–28. [CrossRef]
5. Chung, W.C.; Pestana, G.R.; Kim, M. Structural health monitoring for TLP-FOWT (floating offshore wind turbine) tendon using sensors. *Appl. Ocean. Res.* **2021**, *113*, 102740. [CrossRef]
6. Mieloszyk, M.; Majewska, K.; Ostachowicz, W. Application of embedded fiber Bragg grating sensors for structural health monitoring of complex composite structures for marine applications. *Mar. Struct.* **2021**, *76*, 102903. [CrossRef]
7. Karvelis, P.; Georgoulas, G.; Kappatos, V.; Stylios, C. Deep machine learning for structural health monitoring on ship hulls using acoustic emission method. *Ships Offshore Struct.* **2021**, *16*, 440–448. [CrossRef]
8. Silionis, N.E.; Anyfantis, K.N. Static strain-based identification of extensive damages in thin-walled structures. *Struct. Health Monit.* **2022**, *21*, 2026–2047. [CrossRef]
9. Arrichiello, V.; Gualeni, P. Systems engineering and digital twin: A vision for the future of cruise ships design, production and operations. *Int. J. Interact. Des. Manuf.* **2020**, *14*, 115–122. [CrossRef]
10. Ibrion, M.; Paltrinieri, N.; Nejad, A.R. Learning from failures in cruise ship industry: The blackout of Viking Sky in Hustadvika, Norway. *Eng. Fail. Anal.* **2021**, *125*, 105355. [CrossRef]

11. Woolley, A.; Whitehouse, T. A modelling and simulation framework to assess integrated survivability of naval platforms in high threat environments. *Ocean. Eng.* **2022**, *257*, 111479. [CrossRef]
12. Mauro, F.; Kana, A.A. Digital twin for ship life-cycle: A critical systematic review. *Ocean. Eng.* **2022**, *269*, 113479. [CrossRef]
13. Ludvigsen, K.B.; Smogeli, Ø. *Digital Twins for Blue Denmark*. Marine Cybernetics Advisory; DNV GL Maritime: Hamburg, Germany, 2018.
14. Coraddu, A.; Oneto, L.; Baldi, F.; Cipollini, F.; Atlar, M.; Savio, S. Data driven ship digital twin for estimating the speed loss caused by the marine fouling. *Ocean. Eng.* **2019**, *186*, 106063. [CrossRef]
15. Tofte, B.L.; Vennemann, O.; Mitchell, F.; Millington, N.; McGuire, L. How digital technology and standardisation can improve offshore operations. In Proceedings of the Offshore Technology Conference, Offshore Technology Conference, Houston, TX, USA, 6–9 May 2019.
16. Bernasconi, A.; Kharshiduzzaman, M.; Anodio, L.F.; Bordegoni, M.; Re, G.M.; Braghin, F.; Comolli, L. Development of a monitoring system for crack growth in bonded single-lap joints based on the strain field and visualization by augmented reality. *J. Adhes.* **2014**, *90*, 496–510. [CrossRef]
17. Schirmann, M.; Collette, M.; Gose, J. Ship motion and fatigue damage estimation via a digital twin. In *Proceedings of the 6th International Symposium on Life-Cycle Civil Engineering (IALCCE)*, Ghent, Belgium, 28–31 October 2018; Caspee, R., Taerwe, L., Frangopol, D.M., Eds.; CRC Press: Boca Raton, FL, USA, 2018; pp. 2075–2082.
18. Lloyds Register. *ShipRight Ship Event Analysis*; Lloyds Register: London, UK, 2004.
19. American Bureau of Shipping. *American Bureau of Shipping Guide for Hull Condition Monitoring Systems*; American Bureau of Shipping: Houston, TX, USA, 1995.
20. Majewska, K.; Mieloszyk, M.; Ostachowicz, W.; Król, A. Experimental method of strain/stress measurements on tall sailing ships using Fibre Bragg Grating sensors. *Appl. Ocean. Res.* **2014**, *47*, 270–283. [CrossRef]
21. Tessler, A. *A Variational Principle for Reconstruction of Elastic Deformations in Shear Deformable Plates and Shells*; National Aeronautics and Space Administration, Langley Research Center: Hampton, VA, USA, 2003.
22. Tessler, A.; Spangler, J.L. Inverse FEM for full-field reconstruction of elastic deformations in shear deformable plates and shells. In Proceedings of the 2nd European Workshop on Structural Health Monitoring, Munich, Germany, 7–9 July 2004.
23. Gherlone, M.; Cerracchio, P.; Mattone, M.; Di Sciua, M.; Tessler, A. Shape sensing of 3D frame structures using an inverse finite element method. *Int. J. Solids Struct.* **2012**, *49*, 3100–3112. [CrossRef]
24. Gherlone, M.; Cerracchio, P.; Mattone, M.; Di Sciua, M.; Tessler, A. An inverse finite element method for beam shape sensing: Theoretical framework and experimental validation. *Smart Mater. Struct.* **2014**, *23*, 045027. [CrossRef]
25. Wang, Y.; Nnaji, B.O. Document-driven design for distributed CAD services in service-oriented architecture. *J. Comput. Inf. Sci. Eng.* **2006**, *6*, 127–138. [CrossRef]
26. Freeman, I.; Salmon, J.; Coburn, J. A bi-directional interface for improved interaction with engineering models in virtual reality design reviews. *Int. J. Interact. Des. Manuf.* **2018**, *12*, 549–560. [CrossRef]
27. Kefal, A.; Oterkus, E. Displacement and stress monitoring of a chemical tanker based on inverse finite element method. *Ocean. Eng.* **2016**, *112*, 33–46. [CrossRef]
28. CEI—Creators of Enight Visualization Software. 2011. Available online: <http://www.ensight.com/> (accessed on 29 November 2011).
29. VTFx Format—Ceetron. 2011. Available online: <http://www.ceetron.com/vtfx%20format.aspx> (accessed on 29 November 2011).
30. Song, I.-H.; Yang, J.; Jo, H.; Choi, S. Development of a lightweight CAE middleware for CAE data exchange. *Int. J. Comput. Integ. M.* **2001**, *22*, 823–835. [CrossRef]
31. Cho, S.W.; Kim, S.W.; Park, J.-P.; Yang, S.W.; Choi, Y. Engineering collaboration framework with CAE analysis data. *Int. J. Precis. Eng. Man.* **2011**, *12*, 635–641. [CrossRef]
32. Liu, Q.; Li, J.; Liu, J. ParaView visualization of Abaqus output on the mechanical deformation of complex microstructures. *Comput. Geosci.* **2017**, *99*, 135–144. [CrossRef]
33. Hambli, R.; Chamekh, A.; Salah HB, H. Real-time deformation of structure using finite element and neural networks in virtual reality applications. *Finite Elem. Anal. Des.* **2006**, *42*, 985–991. [CrossRef]
34. Connell, M.; Tullberg, O. A framework for immersive FEM visualization using transparent object communication in a distributed network environment. *Adv. Eng. Softw.* **2002**, *33*, 453–459. [CrossRef]
35. Tzong-Ming, C.; Tu, T.H. A fast parametric deformation mechanism for virtual reality applications. *Comput. Ind. En.* **2009**, *57*, 520–538. [CrossRef]
36. Ryken, M.J.; Vance, J.M. Applying virtual reality techniques to the interactive stress analysis of a tractor lift arm. *Finite Elem. Anal. Des.* **2000**, *35*, 141–155. [CrossRef]
37. Lee, J.H.; Nam, Y.S.; Kim, Y.; Liu, Y.; Lee, J.; Yang, H. Real-time digital twin for ship operation in waves. *Ocean. Eng.* **2022**, *266*, 112867. [CrossRef]
38. Assani, N.; Matic, P.; Katalinić, M. Ship's digital twin—A review of modelling challenges and applications. *Appl. Sci.* **2022**, *12*, 6039. [CrossRef]
39. Tessler, A.; Spangler, J.L. A least-squares variational method for full-field reconstruction of elastic deformations in shear-deformable plates and shells. *Comput. Methods Appl. Mech. Eng.* **2005**, *194*, 327–339. [CrossRef]

40. Maleshkov, S.; Chotrov, D. Post-processing of engineering analysis results for visualization in VR system. *Int. J. Comput. Sci. Issues* **2013**, *10*, 258–263.
41. Rodriguez, I. *Dynamic Occlusion Culling Using Octrees with Unity for Virtual Reality*; The University of Texas at San Antonio: San Antonio, TX, USA, 2017.
42. Chen, L.; Xu, J. Optimal delaunay triangulations. *J. Comput. Math.* **2004**, *22*, 299–308.
43. Woo, Y. Abstraction of mid-surfaces from solid models of thin-walled parts: A divide-and-conquer approach. *Comput-Aided Des.* **2014**, *47*, 1–11. [CrossRef]
44. Yeh, T.P.; Vance, J.M. Applying virtual reality techniques to sensitivity-based structural shape design. *J. Mech. Des.* **1998**, *120*, 612–619. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Fully Buried Pipeline Floatation in Poro-Elastoplastic Seabed under Combined Wave and Current Loadings

Jian Leng ¹, Libin Liu ¹, Chencong Liao ^{2,*} and Guanlin Ye ²

¹ CCCC Third Harbor Engineering Co., Ltd., Shanghai 200030, China; keepwalkinglj@163.com (J.L.); liulibin@ccccltd.cn (L.L.)

² State Key Laboratory of Ocean Engineering, Department of Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China; ygl@sjtu.edu.cn

* Correspondence: billaday@sjtu.edu.cn; Tel.: +86-135-6479-6195

Abstract: The floatation capacity of seabed pipelines has long been considered a key risk element during design, especially with the combined loading of waves and currents. This paper presents a two-dimensional coupled approach with a poro-elastoplastic theory to study the floatation of pipelines with the combined loading of waves and currents. The findings suggest that the proposed method is able to capture the mechanical performance of pipeline floatation. Pipeline floatation occurs in two distinct phases. In the initial phases, the pipelines float slowly with the cyclic loadings. In the second stage, when the backfill soil in the middle position of the pipelines begins to liquefy, the floating displacement increases obviously. The boundary constraints provided by the pipelines strengthen the backfill soil as well as accelerate the release of excessive pore water pressure. Meanwhile, a nonliquefiable region is formed under the pipelines. The floating displacement of the pipelines increases as well as current velocity, wave height, and wave period, and reduces with increased backfill soil permeability. Increasing the permeability coefficient of backfill soil can obviously restrain the floatation of pipelines.

Keywords: seabed pipeline; wave and current; excess pore pressure; porous media; liquefaction; effective stress

1. Introduction

Seabed pipelines have a common application in oil and gas transportation. Seabed pipelines are usually fully buried offshore while partially buried deep in sea. However, wave and current loadings induce an increase in excess pore water pressure within the seabed soil, thereby reducing its effective stress. The shallow seabed will even liquefy. In the liquefied seabed, it will float up without weighting measures. The floatation of the pipeline will destroy the overall stability and ultimately affect the safety of the pipelines. The floatation capacity of seabed pipelines has long been considered a key risk element, and studying pipeline floatation is crucial for the checking procedures of subsea pipelines [1,2]. Therefore, to ensure seabed pipeline safety, it is essential to assess the dynamic reaction of the seabed pipeline to combined wave and current loadings, particularly when the pipeline floats.

The use of coupled models that incorporate poro-elastoplastic theory has greatly increased the understanding of seabed pipelines floatation [3–5]. For instance, Qi and Gao [6] and Liang et al. [7] developed models that account for fluid–seabed–pipeline interactions, capturing the effects of cyclic wave loading and soil liquefaction. Numerous researchers have employed numerical modeling techniques to examine the soil–structure interaction of marine constructions [8–10]. Based on the literature review above, studies treating the sandy seabed as an elastoplastic material remain very limited at present, because the oscillatory response can be described accurately, while the residual pore

pressure cannot be captured. In addition to numerical analysis, both oscillatory mechanism and residual liquefaction have been observed in laboratory experiments [11–13]. The wave flume test is a common method that can intuitively simulate wave conditions. The seabed reaction to wave action was reported by Zhang et al. [14], Wang et al. [15], and Smith et al. [16], respectively, using wave flume testing. In addition to wave flume test, Smith et al. [17], Jones et al. [18], and Wang et al. [19] elucidated the dynamic behavior of seabed particles under flow conditions through centrifuge experiments and numerical simulations. Due to the limitations of the laboratory equipment, only the pressure of pore water can be determined due to equipment restrictions in the laboratory. However, numerical approximations can effectively compensate for these deficiencies.

The situation becomes more complex when the fluid–seabed interaction takes a pipeline into account. However, since the 1980s, the interaction between waves, the seabed, and pipelines has been the subject of numerous investigations [20,21]. In recent years, Zhang et al. [22] introduced a 3D buried seabed pipelines simulation using the existing DYNE3WAC system as a basis. Building upon the assumption of poro-elastic soil material and the suggested numerical model, the implications of soil and wave features on the seabed reaction around the seabed pipelines induced by waves were examined. Zhao et al. [23] considered the preconsolidation caused by the pipeline’s weight and proposed one model that included a new definition of residual pore pressure generations. Zhao’s study investigated the impacts of the backfill depth on the soil dynamic reaction surrounding the pipeline, but the effects of current loadings were still ignored. Zhao et al. [24] used a two-dimensional numerical framework to explore how cross-anisotropic soil properties influence the dynamic seabed reaction and presented a simpler approximation for determining max liquefied depth based on comprehensive parametric studies. It is worth noting that the earlier research exclusively examined the impact of wave loading on the dynamic response of the seabed. In recent years, although there have been many excellent studies on floatation of seabed pipelines exposed to complicated wave and current loadings [25,26], the majority of soil constitutive models employed are static, with relatively few studies focusing on dynamic soil models.

This paper presents a two-dimensional coupled approach with a poro-elastoplastic theory to study the floatation of pipelines with the combined loading of waves and currents. The cyclic constitutive model [27,28] incorporates the structural, overconsolidation, and anisotropic characteristics of soil, facilitating an accurate portrayal of the static and dynamic behaviors of both sandy and clayey soils, rendering it more comprehensive compared to the elastoplastic model. It can more accurately simulate the dynamic response of pipelines under wave and current loadings, particularly when the pipeline floats. A comprehensive discussion of the results is presented, involving those on the mechanism of pipeline floatation and the impacts of current and wave characteristics, and particular attention is paid to the impact of soil permeability.

2. Materials and Methods

2.1. Wave Module

To accurately simulate the dynamic response of the seabed, this paper employed the third-level approximation wave–current interaction model widely used in ocean engineering research. This model could better capture the complex interactions between waves and currents. By integrating the dispersion relation, it enhances the ability to forecast wave behavior across diverse water depth scenarios, thereby ensuring versatility in its practical application, and it can comprehensively calculate the dynamic pressure on the seabed. Below is a detailed introduction about the model.

Hsu et al. [29] assumed that sea water is an in-viscous and incompressible fluid and proposed a third-level estimate for the interaction between waves and currents. The corresponding relationship is outlined as follows:

$$\phi(x, z, t) = -U_0 x + \frac{Hg \cosh \kappa}{2(U_0 \kappa - \omega_0) \cosh \kappa d} \sin(\kappa x - \omega t) + \frac{3H^2 \cosh 2\kappa(z-h)}{32(\sinh \kappa d)^4} (U_0 - \omega_0) \sin 2(\kappa x - \omega t) + \frac{3\kappa^3 H^3}{512} \frac{(9-4(\sinh \kappa d)^2) \cosh(\kappa d)}{(\sinh \kappa d)^3} (U_0 - \omega_0) \sin 3(\kappa x - \omega t) \quad (1)$$

$$\eta(x, t) = \frac{H}{2} \cos(\kappa x - \omega t) + \frac{\kappa H^2}{16} \frac{[3+2(\sinh \kappa d)^2] \cosh(\kappa d)}{(\sinh \kappa d)^3} \cos 2(\kappa x - \omega t) + \frac{\kappa^2 H^3}{512} \frac{[3+14(\sinh \kappa d)^2+2(\sinh \kappa d)^4]}{(\sinh \kappa d)^4} \cos(\kappa x - \omega t) + \frac{\kappa^2 H^3}{512} \frac{3*[9+24(\sinh \kappa d)^2+24(\sinh \kappa d)^4+8(\sinh \kappa d)^6]}{(\sinh \kappa d)^4} \cos 3(\kappa x - \omega t) \quad (2)$$

$$C(t) = \frac{U_0^2}{2} - \frac{H^2}{16} \frac{(\omega_0 - U_0 \kappa)^2}{(\sinh \kappa d)^2} \quad (3)$$

where Φ is the velocity potential, g represents the gravitational acceleration of the earth, k signifies the wave vector, U_0 denotes the flow velocity, H stands for the first-order wave height, $C(t)$ represents Bernoulli's constant, d indicates the water depth, and η refers to the free surface elevation relative to the static water level.

The dispersion relationship can be stated as

$$\omega = \omega_0 + (\kappa H)^2 \omega_2 \quad (4)$$

$$\omega_0 = U_0 \kappa + \sqrt{g \kappa \tanh \kappa d} \quad (5)$$

$$\omega_2 = \frac{[9 + 8(\sinh \kappa d)^2 + 8(\sinh \kappa d)^4]}{64(\sinh \kappa d)^4} (\omega_0 - U_0 \kappa) \quad (6)$$

It can be seen that when $U_0 = 0$, the above formula represents the traditional solution with no currents. Further, Ye and Jeng [30] proposed the dynamic pressure that is applied to the seabed when it is paired with current and wave loadings:

$$P_b(x, t) = \frac{\rho_f g H}{2 \cosh \kappa d} \left[1 - \frac{\omega_2 \kappa^2 H^2}{2(U_0 \kappa - \omega_0)} \right] \cos(\kappa x - \omega t) + \frac{3\rho_f \kappa H^2}{8} \left[\frac{\omega_0(\omega_0 - U_0 \kappa)}{2(\sinh \kappa d)^4} - \frac{g \kappa}{3 \sinh 2 \kappa d} \right] \cos 2(\kappa x - \omega t) + \frac{3\rho_f \kappa H^3 \omega_0(\omega_0 - U_0 \kappa)}{512} \frac{(9-4(\sinh \kappa d)^2)}{(\sinh \kappa d)^7} \cos 3(\kappa x - \omega t) = P_1 \cos(\kappa x - \omega t) + P_2 \cos 2(\kappa x - \omega t) + P_3 \cos 3(\kappa x - \omega t) \quad (7)$$

2.2. Seabed Module

Particle shape, density, stress and strain history, structure, and overconsolidation all significantly affect the mechanical properties of sandy seabeds. In addition, with different loading and drained conditions, the mechanical performance of sand is completely different. For example, loose sand is going to liquefy under cyclic loadings in undrained situations. In contrast, dense sand will not liquefy. Under drained conditions, loose sand will be compacted and not liquefied. Zhang et al. [27,28] and Ye et al. [31] suggested a cyclic mobility constitutive model based on notions of subloading [32] and superloading [33]. The suggested model is able to represent the physical properties of drained and undrained sand and clay under monotonic and dynamic loadings, as well as medium-density sand and clay cyclic mobility.

This cyclic mobility constitutive model contains eight parameters, with b_r , a , and m regulating the development rate of anisotropy, the structural loss rate, and the decay rate for overconsolidation, respectively. The remaining five parameters are identical to those in the Cam–Clay model. The physical interpretation of the three parameters is evident, so they can be determined easily by laboratory experiments [27,28,31,34,35]. The model was used in numerical assessments of various dynamic reactions [28,31,34–39].

2.3. Numerical Formulation

A two-phase theory fully coupled with u - p formulation [40,41] is applied in dynamical finite element valuation of sandy seabeds. In saturated soil, pore water pressure is denoted as p and soil deformation as u . The u - p model assumes infinitesimal strain, minor temporal and spatial variations in void ratio relative to other variables, negligible relative acceleration between soil and pore water compared to within the soil, and incompressible soil particles. Equations (8) and (9), respectively, represent the continuity equation and the equilibrium equation.

$$\rho \ddot{u}_i = \frac{\partial \sigma_{ij}}{\partial x_j} + \rho b_i \quad (8)$$

$$\rho^f \ddot{\varepsilon}_{ii}^s - \frac{\partial^2 p_d}{\partial x_i \partial x_i} - \frac{\gamma_w}{k} \left(\frac{\dot{\varepsilon}_{ii}^s}{n} - \frac{1}{K_f} \dot{p}_d \right) = 0 \quad (9)$$

where ρ represents the soil density, u denotes the soil displacement, ρ^f stands for fluid density, ε_{ii}^s is the volumetric strain of soil skeleton, p_d denotes the pore water pressure, γ_w represents the unit weight of the pore fluid, k stands for the permeability coefficient, K_f stands for the volumetric compressibility of the fluid, and n denotes the volumetric compressibility of the soil porosity.

Based on the above u - p fully coupled dynamic analysis theory, in this paper, a hybrid scheme of finite element and finite difference (FE-FD) is employed for discretizing Equations (8) and (9) in space. Under the (FE-FD) hybrid scheme, the displacements of solids are discretized using the finite element approach, and the excess pore pressures of liquids are calculated using the finite difference approach. Specific formulation and the validation of accuracy can be obtained in Jeng and Ou [42]. On the basis of the FE-FD hybrid scheme, Ye et al. [35] created a program called DBLEAVES, which conducts finite element analysis in both 2D and 3D for static and dynamic scenarios. Subsequently, numerous studies have confirmed the program's precision and usefulness [34,37–39,43].

2.4. Model Validation

In this part, the experimental findings of Sumer et al.'s reservoir trial [12] are used to validate the model's accuracy and numerical analytic methodologies. Figure 1 depicts the modeling and experimental results of EPWP (excessive pore water pressure at three depths, $Z = 0$ cm, 7.5 cm, and 12.5 cm). It can be shown that the mathematical findings match the tests in terms of EPWP buildup at various depths as well as EPWP vibration following liquefaction. From a holistic perspective, this model effectively simulates the trends and magnitudes of experimental data, exhibiting high accuracy. The differences between simulated results and experimental data, particularly at depths of 7.5 cm and 12.5 cm, are shown in Figure 1. The main reasons include the following: The wave generator cannot produce ideal regular waves. Model experiments are subject to boundary effects, and currently, these effects cannot be completely eliminated. Boundary effects are also difficult to quantify. In this finite element study, the model size reaches 100 m with the aim of minimizing the influence of boundary effects as much as possible.

2.5. FEM Model and Boundary Assumptions

Figure 2 illustrates the seabed-pipeline model with a single layer of sandy seabed. The 2D model presents dimensions of 12 m in height and 100 m in width. The boundaries on the right and left are constrained in the X-axis direction and are limitless along the Z-axis. The lower boundary is secured in both the X and Z directions. Figure 3 depicts the maximum excess pore pressures induced by waves as well as currents on the seabed element positioned at the pipeline base, analyzed across various mesh mechanisms denoted by the entire amount of elements N , in which u_e represents maximum excess pore pressure and σ'_0 signifies beginning effective stress. The outcomes depicted in Figure 3 showcase that the mesh system attains a commendable level of computational precision.

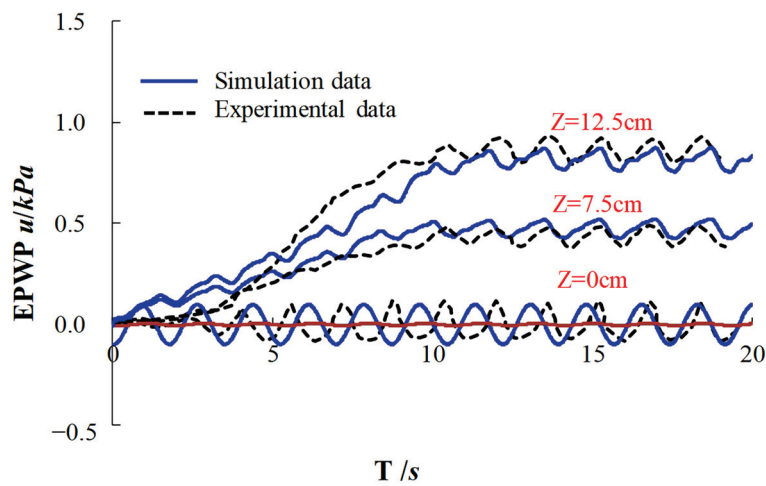


Figure 1. Comparison of experimental data from Sumer et al. [10] with FEM simulated outcomes.

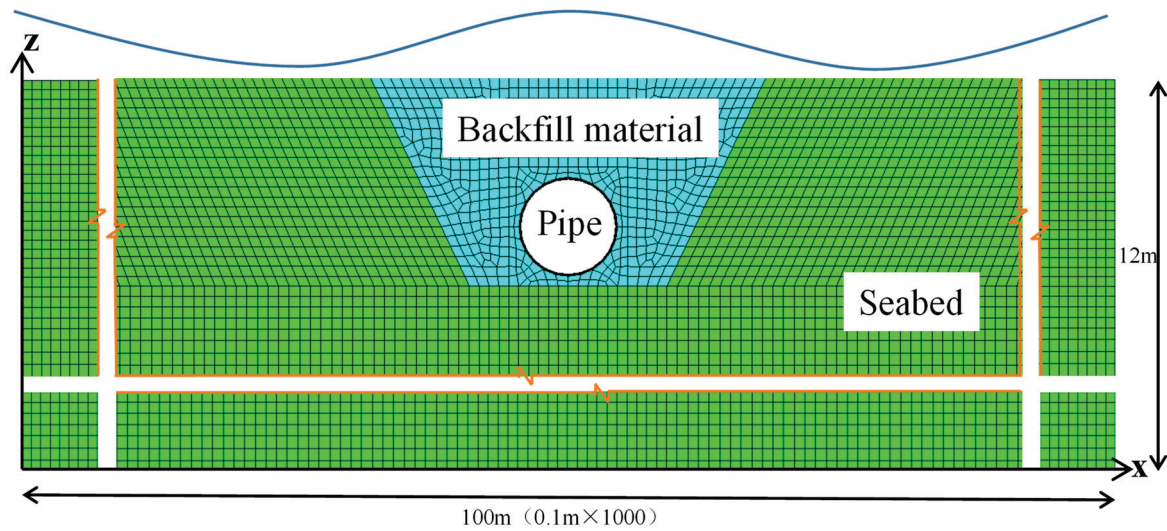


Figure 2. Mesh domain for the seabed-pipeline FEM model.

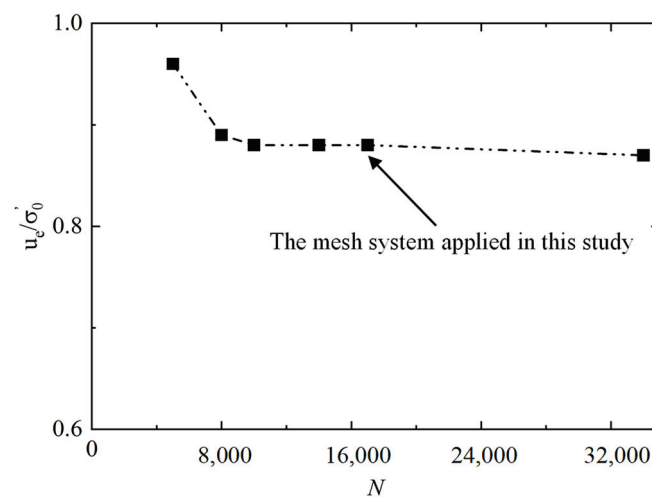


Figure 3. The ratio (u_e/σ'_0) of maximum excess pore pressure generated by waves and currents at the pipeline's underside for diverse mesh frameworks.

Fully buried pipelines are typically distributed along the coast. Therefore, this study employed actual observed wave data characteristic of ocean stations. The specific load parameters are as follows: wave height $H = 1$ m; water depth $d = 10$ m; wave period $T = 5$ s; the velocity of the current $u = 1$ m/s. As obtaining soil samples offshore is very challenging, the soil parameters in this study were chosen from widely cited standard sand parameters in the literature [28]. The Toyoura sand, which is widely applied to the liquefaction related model and element tests, simulates the sandy seabed. The parameters of the backfill soil have the same properties as the original seabed soil, except for the permeability coefficient. Table 1 displays the input parameters for numerical modeling, unless otherwise indicated. The pipeline parameters were selected based on actual engineering cases. The density of the steel pipeline is $\rho = 7.85 \times 10^3$ kg/m³ and the pipeline is assumed to be filled with water ($\rho = 1.0 \times 10^3$ kg/m³). The specific gravity of pipes included in the contents (the ratio of the gravity of pipes to the gravity of water) is 1.51, while the specific gravity of the seabed is 1.9.

Table 1. Parameters for numerical simulation.

Parameter	Value	Unit
Seabed Parameters		
Compression index λ	0.05	
Swelling index k	0.0064	
Relative density D_r	62%	
Permeability coefficient k_s	10^{-7}	m/s
Poisson's ratio ν	0.30	
Critical state parameter M	1.2	
Void ratio N ($p' = 98$ kPa on N.C.L)	0.74	
Evolution parameter of anisotropy b_r	1.5	
Degradation parameter of overconsolidation state m_R	0.1	
Degradation of structure m_R^*	2.2	
Initial degree of structure R_0^*	0.8	
Initial consolidation ratio $1/R_0$	25	
Initial anisotropy ζ_0	0.0	
Backfill Parameters		
Permeability coefficient k_s	10^{-6}	m/s
Bottom of the trench width W_1	2	m
Top of the trench width W_2	4	m
Trench height h	2.1	m
Distance between seabed pipeline and bottom of trench d	0.1	m
Wave and Current Parameters		
Current velocity u	+1	m/s
Wave period T	5	s
Water depth D	10	m
Water height H	1	m
Pipeline Parameters		
Pipeline thickness t	0.2	m
Outer diameter D	0.5	m
Young's modulus E_p	2.09×10^{11}	N/m ²

3. Results and Discussion

3.1. Floatation of Pipeline and Development of EPWPR in Seabed

The node displacement at the bottom of the simulated pipeline is used as the analysis object to analyze the floatation of pipelines, as pipelines could be treated as rigid bodies compared with seabed soil. Figure 4 depicts the pipeline's floating displacement over time. It is evident that when combined wave and current loadings are applied, the pipeline

floats and oscillates. For the convenience of subsequent analysis, the midpoint of the displacement of each oscillation cycle is taken as the pipe displacement in one cycle. There are two stages to pipeline floatation. During the initial stage, the pipeline floats slowly with the cyclic loading. When the loading time reaches 350 s, the pipeline floating displacement reaches the inflection point and enters the second stage, and the floating displacement increases rapidly. This agrees with results of wave flume experiments by Sumer et al. [44], and existing experiments show that pipelines will eventually float to the seabed surface. According to Figure 5, the displacement vectors of the seabed are changed by the pipeline. The displacement vectors of the soil above the seabed pipeline exhibit significant undulations, which indicates substantial influence by wave-induced dynamic loads. The soil located in front of and beneath the pipeline exerts an upward-angled thrust on the pipeline. The overall horizontal displacement direction aligns with the water flow direction.

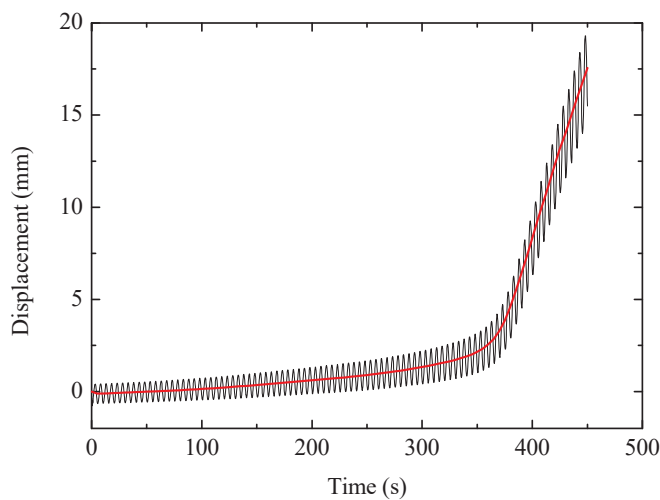


Figure 4. Time history of floating displacement of the pipeline.

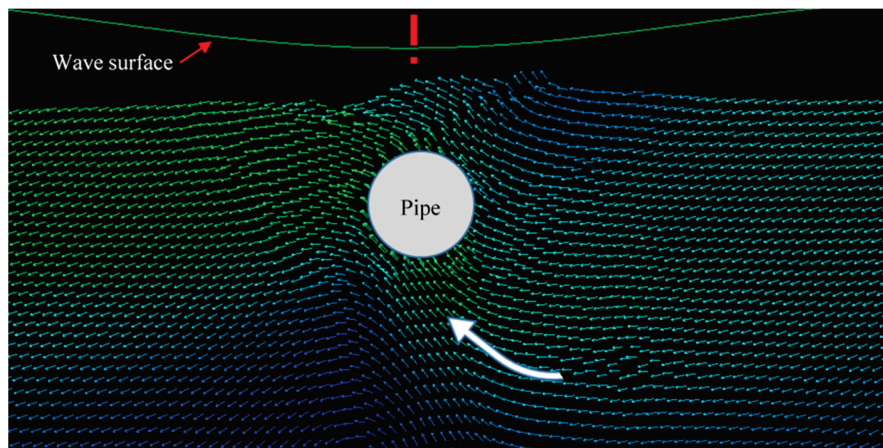


Figure 5. Displacement vectors of the seabed.

By analyzing the mechanical equilibrium conditions of fully buried pipelines, it is determined that the buoyant force acting on the pipeline is equal to the sum of the pipeline's weight and the downward restraining force exerted by the soil. Since the buoyant force and the weight remain constant, when the soil's restraining force on the pipeline decreases, the pipeline will undergo buoyancy. Under cyclic dynamic loading, the seabed undergoes liquefaction, resulting in a decrease in its density and shear strength, thereby causing a reduction in the restraining force exerted by the seabed on the pipeline. To analyze the

reason for the floatation inflection, it is necessary to conduct research on the liquefaction around the pipeline. In this paper, EPWPR (excess pore water pressure ratio) is utilized to denote liquefaction levels. In general, considering both numerical stability and generality, $EPWPR \geq 0.95$ is applied as the liquefaction standard.

Figure 6 depicts the evolution of the liquefaction area. It is evident that the direction of soil liquefaction around the seabed pipeline is predominantly downward. The liquefaction rate of the soil above the seabed pipeline is greater than that of the soil below it. Furthermore, for loading times less than 300 s, the soil beneath the seabed pipeline does not exhibit liquefaction. This conclusion is consistent with the observations shown in Figure 4. Since the permeability coefficient of the soil used for backfill in the trench is larger than that of the seabed, the degree of liquefaction of the trench at the same depth is smaller than that of the seabed. The degree of liquefaction above the pipeline surpasses that observed on its sides, primarily attributed to the impermeable and rigid characteristics of the pipeline. At the same time, a hard-to-liquefy area is formed under the pipeline. The area is approximately 45 degrees around the midpoint of the pipeline. The EPWPR of the soil under the pipeline is notably lower compared to that of the seabed at equivalent depths and under the same period. With the further action of the wave and current loadings, all the soil around the pipeline finally liquefies.

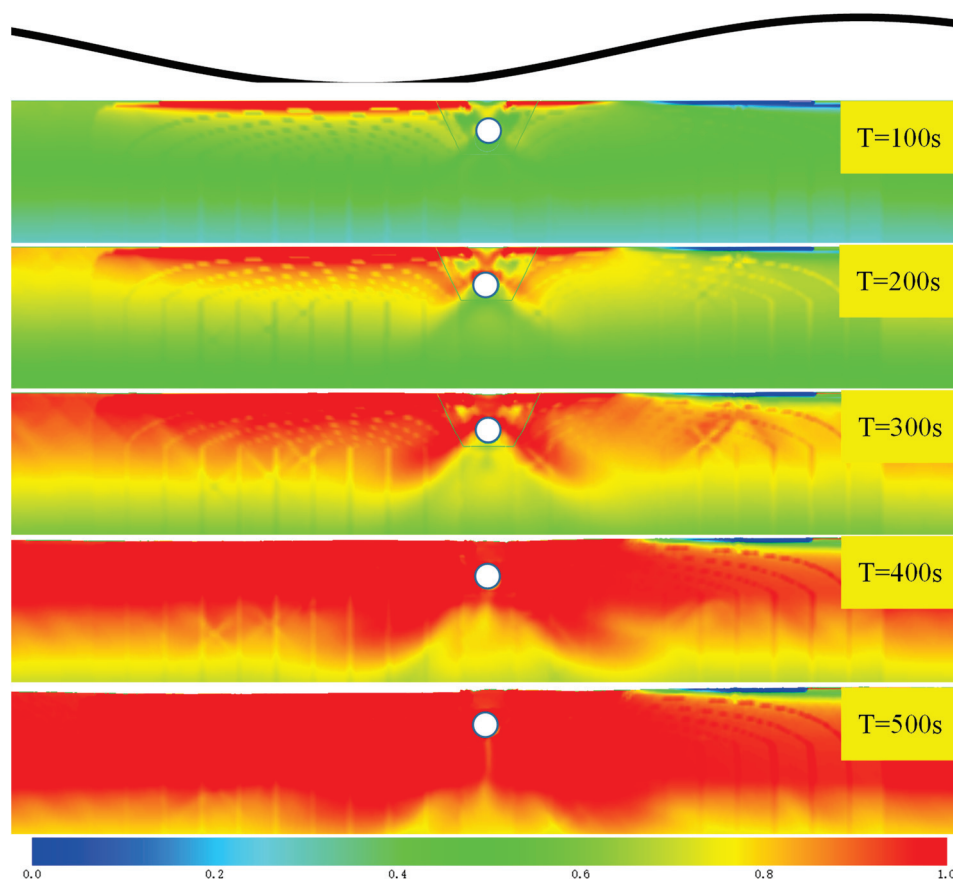
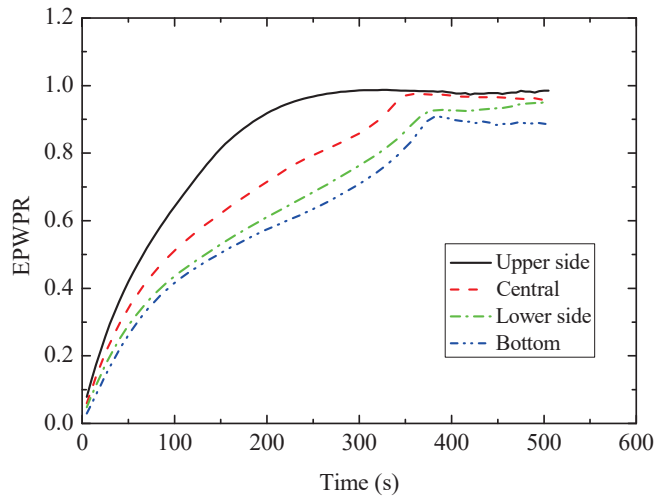


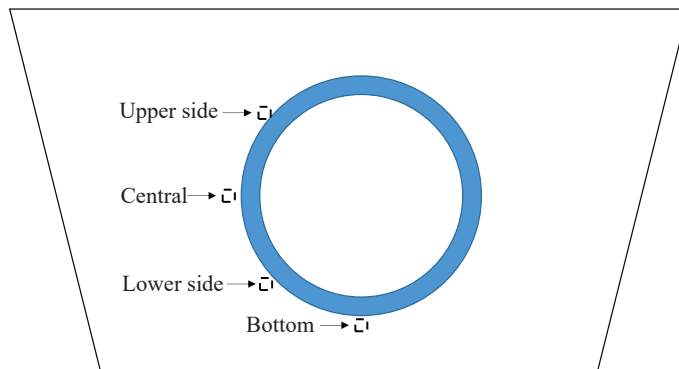
Figure 6. Evolution of soil liquefaction and pipeline flotation over time. 100 s: Surface layer of trench soil begins to liquefy. 200 s: Liquefaction extends deeper. 300 s: Significant liquefaction around pipeline, extends beneath. 400 s: Soil beneath pipeline liquefies, increasing buoyancy. 500 s: Further liquefaction beneath trench, pronounced pipeline flotation.

Figure 7a summarizes the temporal progression of the EPWPR for soil elements at various positions encircling the pipeline. Positions of these elements are shown in Figure 7b. It can be observed that the upper EPWPR develops the fastest, followed by the central, lower, and bottom sections. As the loading time progresses, the difference in EPWPR

between the upper section and other areas gradually increases. This indicates that the buoyant force exerted on the pipeline continues to increase. At 350 s, the soil in the middle begins to liquefy, and the pressure of the soil on the upper part of the pipeline drops to 0, which causes the pipeline to rise faster. Meanwhile, an inflection point appears on the floating displacement curve of the pipeline. This is because after the pipeline floats, there is a redistribution of stress in the surrounding soil, leading to subsequent stabilization.



(a)



(b)

Figure 7. Time histories of EPWPR of soil elements around the pipeline and the position of the elements analyzed. (a) Time histories of EPWPR. (b) The position of the elements analyzed.

Figure 8 illustrates the effective stress path of seabed elements 10 m from the pipeline and soil elements at different positions around the pipeline. The effective stress path of soil is an important tool in the field of geotechnical engineering for studying the mechanical properties and behavior of soil. By analyzing the effective stress path, engineers can better understand and predict the performance of soil under different stress conditions, thereby guiding the design and construction of actual engineering projects.

The effective stress path of seabed element shows that shear stress σ_{xy} oscillates under cyclic wave–current loadings, and the mean effective stress σ_m reduces. The effective stress path finally attains a stable state called cyclic mobility, which is a characteristic mechanical feature of liquefied soil. Compared with the seabed element, the effective stress path of upper side soil element surrounding the pipe slopes to the right, which means that the mean effective stress σ_m raises the shear stress σ_{xy} under the wave and current loadings. This is caused by boundary constraints provided by the pipeline, which strengthen soil around the pipeline, accelerate the dissipation of excess pore water pressure, and increase σ_m . The effective stress path of the soil element in the lower part of the pipeline is similar to that of the remote seabed element, except for the shape at failure. From the perspective of

shear stiffness, the rightward inclination of the effective stress path in the upper soil element indicates a reduction in shear stiffness. This suggests that the pore water pressure in the upper soil accumulates more rapidly than in the lower soil. Based on the final area and shape of the hysteresis loops in various sections, the inclined and thinner final hysteresis loop area in the upper soil element indicates a greater degree of softening compared to the lower soil. The analysis of the effective stress path, when combined with this study, corroborates the accuracy of the soil dynamic constitutive model selected.

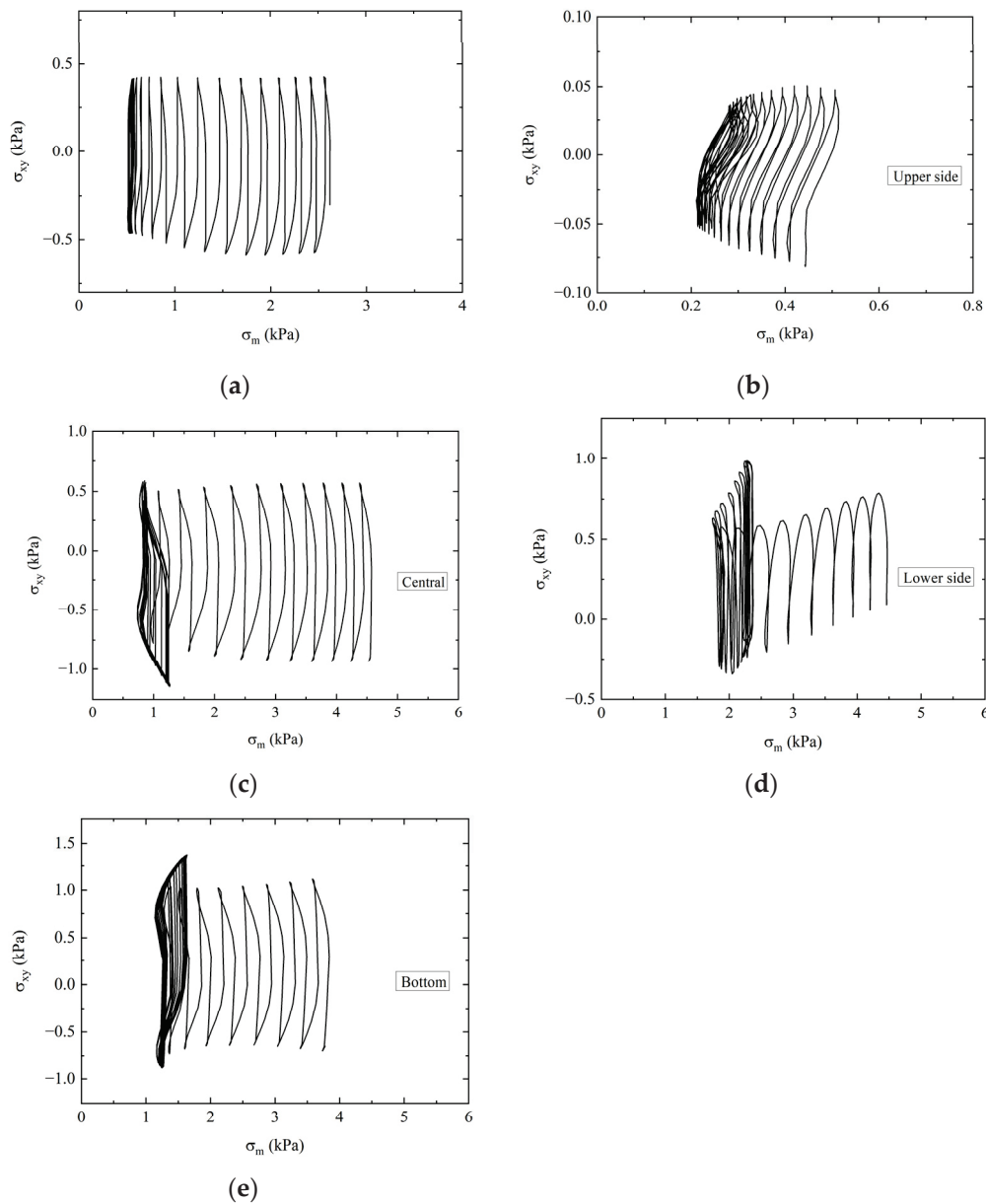


Figure 8. Effective stress path of seabed element 10 m from the pipeline and soil elements at different positions around the pipeline. (a) Effective stress path of seabed element 10 m from the pipeline; (b) Effective stress path of upper side soil element; (c) Effective stress path of central element; (d) Effective stress path of lower side soil element; (e) Effective stress path of bottom soil element.

3.2. Influence of Current and Wave Characteristics

Existing research [5] indicates that a current is crucial to the dynamic response when subjected to the effects of waves and currents. Thereby, three velocities of current ($u = +1$ m/s, 0 m/s, -1 m/s) are picked out to explore the effect of currents on the pipeline's floatation. In the case where $u = 0$, there is no current; for $u < 0$, the current

opposes the direction of the wave; and for $u > 0$, the current aligns with the wave's direction. The temporal records of the pipeline's floating displacement at varying current velocities are presented in Figure 9. All the floatation of the pipeline goes through two stages, the steady rise stage and the accelerated rise stage. According to fluid mechanics theory, the buoyant force exerted by fluid flow on the pipeline is directly proportional to the square of the flow velocity. The higher the fluid flow velocity, the greater the propensity for pipeline buoyancy to occur. The results show that the floating displacement for $u = +1$ m/s is larger than that for $u = -1$ m/s, because the current velocity travelled in the wave's direction increases the wave pressure transmitted to the seabed, accelerating the liquefaction of the soil surrounding the pipeline. As a result, the buoyant force acting on the pipeline increases, leading to flotation. The distributions of the maximum EPWPR below the pipeline (specifically along the vertical line across the center of the pipeline) under various current velocities are shown in Figure 10. The maximum EPWPR of the soil continues to decrease as the depth increases. As the depth of the soil layer increases, the energy transmitted by wave and currents continuously attenuates, consequently diminishing its effect on pore pressure. In addition, the increase in current velocity leads to higher EPWPR, which indicates deeper liquefaction range.

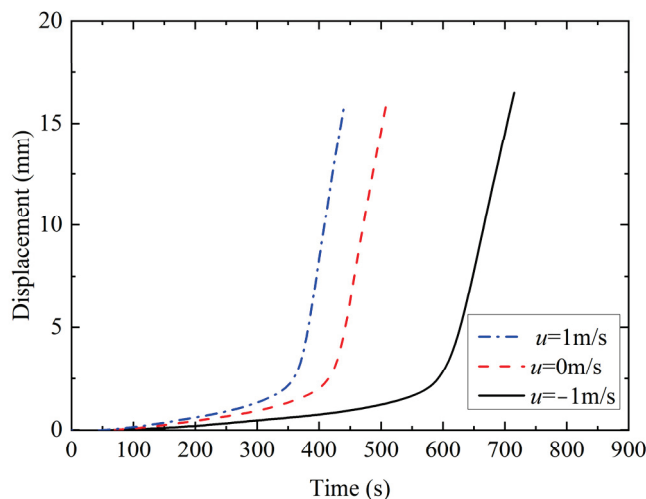


Figure 9. Time histories of floating displacement of pipeline under various current velocities.

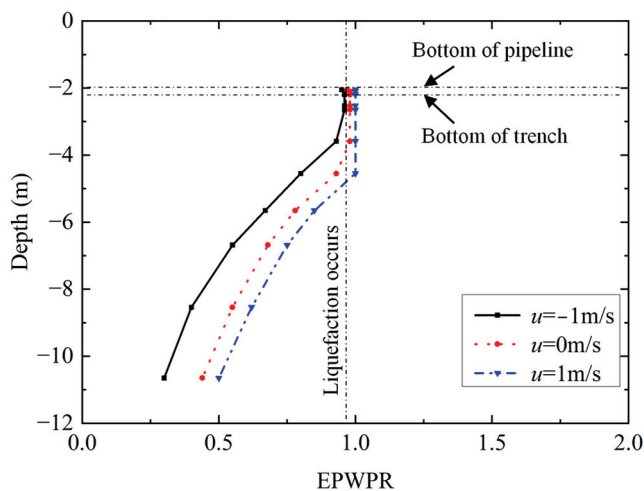


Figure 10. Distributions of the MAX EPWPR along pipeline centerline under current velocities.

Generally, wave characteristics have a considerable influence on seabed dynamic response. Wave period (T) can influence the effective stress and excess pore water pressure

in the seabed by affecting wave length (L), and wave height (H) is able to directly affect the wave pressures on the seabed. Parametric research is performed to examine the impact of wave period (T) on the floatation of pipeline with wave period (T) ranging from 3 s to 6 s with a 1 s gap. Figure 11 shows a visual representation of the pipeline's floating displacement over time and distributions of the maximum EPWPR below the pipeline with different wave periods (T). The findings indicate that an increase in the wave period leads to a rise in the pipeline's floating displacement and, consequently, an increase in the wave pressures exerted on the seabed. As the wave period grows, so too does its influence on the maximum EPWPR. However, as the wave period increases, its effect on the seabed does not proportionally increase. Instead, it undergoes a continuous weakening process. Figure 11b also demonstrates that the maximum EPWPR of the trench soil is lower than that of the natural seabed below the trench with a wave period of three seconds. This is because the permeability of the trench is higher than that of the seabed, indicating that increasing permeability coefficient of the trench can slow down the liquefaction of the soil around the pipelines.

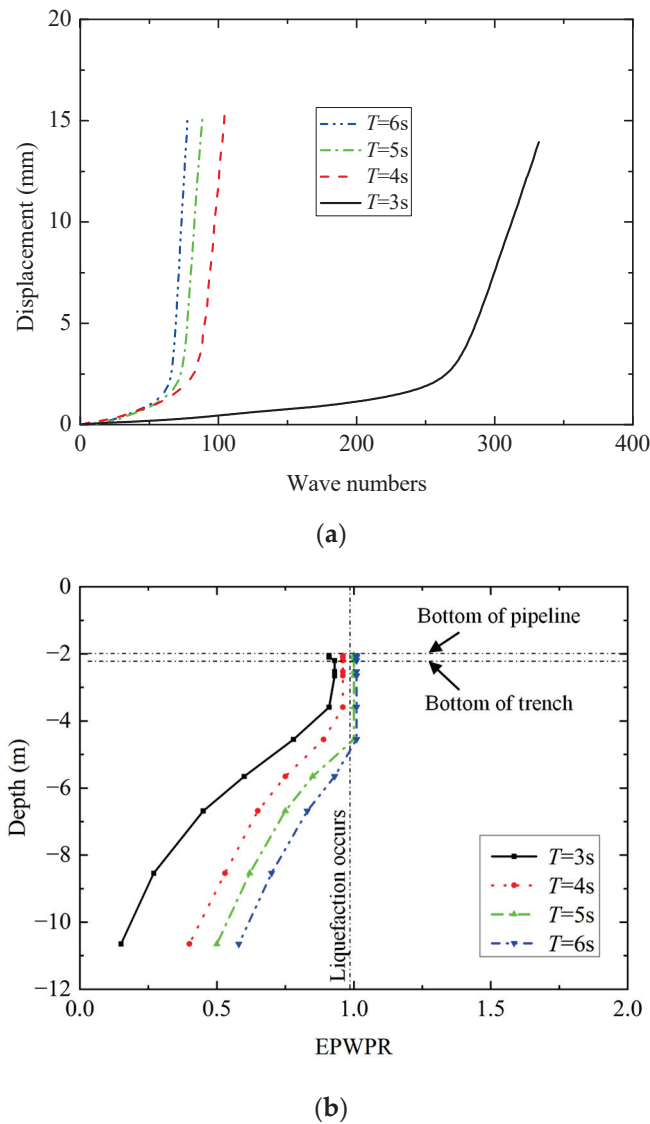


Figure 11. Floating displacement and MAX EPWPR time series for pipelines under various water period. (a) Time histories of floating displacement; (b) Distributions of the MAX EPEPR.

Figure 12 depicts the temporal history of the pipeline floating displacement and distributions of the maximum EPWPR below the pipeline with different wave height ($H = 2.0$ m,

1.5 m, 1.0 m, 0.5 m). As the wave height increases, both the floating displacement of the pipeline and the maximum EPWPR in the seabed increase. And there is not a significant change in pipeline displacement and the maximum EPWPR when the wave height is 1.5 m compared to when it is 2 m. As the wave height continues to increase, its impact on the seabed does not proportionally increase but, rather, undergoes a diminishing effect. When wave height $H = 0.5$ m, which means the wave pressure is the smallest, the highest EPWPR in the trench soil is, likewise, lower than that of the natural seabed below the trench.

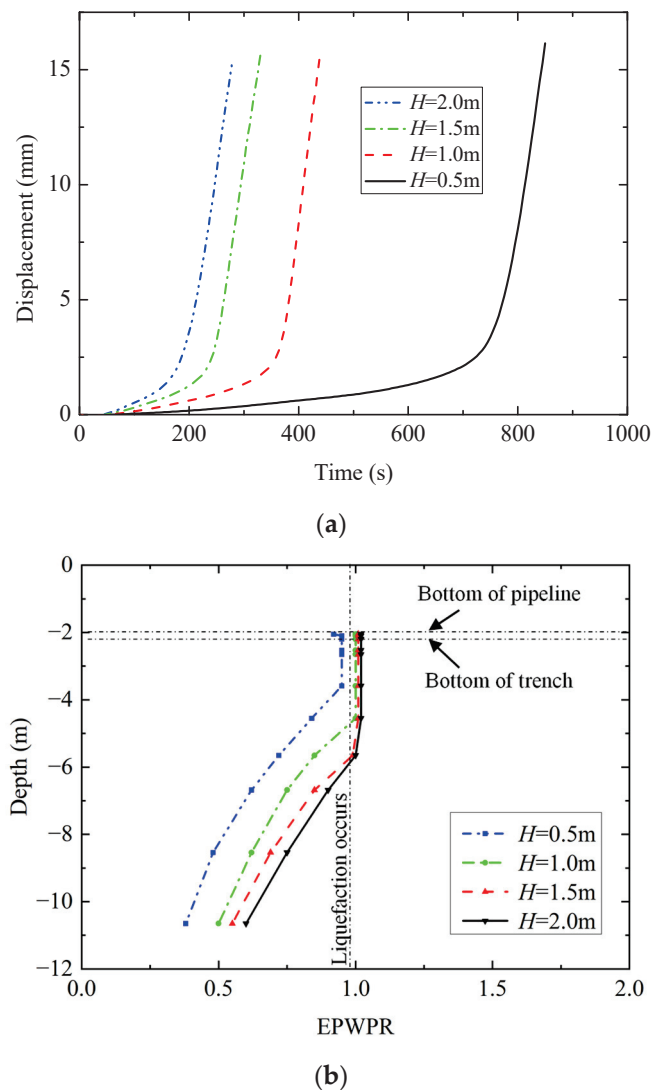


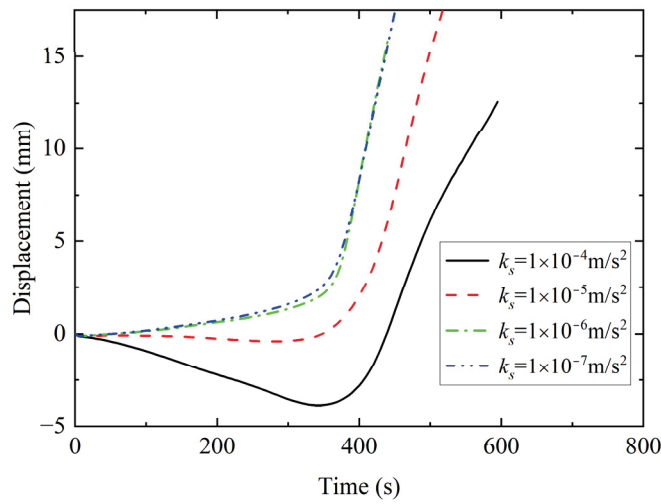
Figure 12. Floating displacement and MAX EPWPR time series for pipelines under various water height. (a) Time histories of floating displacement; (b) Distributions of the MAX EPWPR.

3.3. The Effect of the Backfill Soil's Permeability

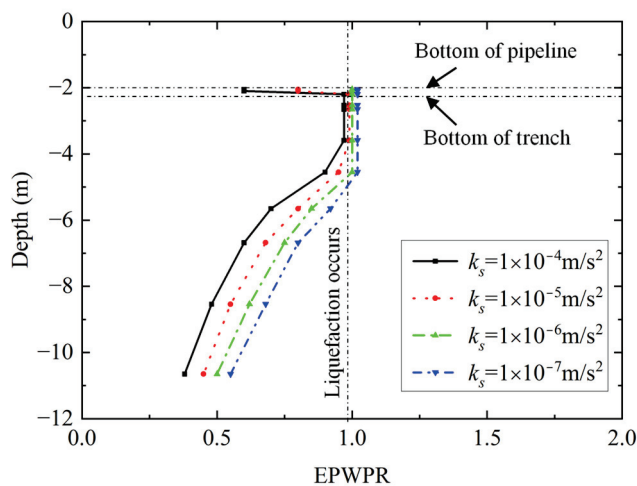
Permeability coefficient stands out as a crucial soil parameter in the seabed dynamic reaction analysis. It is critical to the development of EPWPR, with an important effect on seabed liquefaction. In theory, the higher the permeability coefficient of soil, the faster the dissipation of pore water pressure within the soil, resulting in a slower accumulation of EPWPR and making liquefaction less likely to occur. To explore the impact of permeability coefficient on the floatation of pipelines, various permeability coefficients of the trench soil ($k_s = 1 \times 10^{-4}$ m/s, 1×10^{-5} m/s, 1×10^{-6} m/s, 1×10^{-7} m/s) are examined.

Figure 13a demonstrates how the permeability coefficient impacts the pipeline's buoyancy and the distribution of the highest EPWPR levels below the pipeline. Figure 13b shows, when the permeability coefficient $k_s = 1 \times 10^{-6}$ m/s and 1×10^{-7} m/s, that the

time histories of floatation develop similarly, with a steady increase in the early stage. When the backfill soil in the middle position of the pipeline begins to liquefy, the floating displacement increases rapidly. When the permeability coefficient $k_s = 1 \times 10^{-4} \text{ m/s}$ and $1 \times 10^{-5} \text{ m/s}$, due to the large permeability coefficient of the backfill soil in the trench, the excess pore water pressure dissipates quickly, and the pipeline does not float. In particular, when the permeability coefficient $k_s = 1 \times 10^{-4} \text{ m/s}$, the trench settles down relative to the neighboring seabed, leading in sinking of the pipeline, owing to the large difference in permeability coefficient between the trench soil and seabed. With the cyclic action of the wave and current loadings, the pipe floats rapidly when the natural seabed under the trench liquefies. Figure 13b indicates that the maximum EPWPR decreases, with increasing permeability coefficient of the trench soil. The trench soil will not liquefy when the permeability coefficient $k_s = 1 \times 10^{-4} \text{ m/s}$ and $1 \times 10^{-5} \text{ m/s}$. Therefore, increasing the permeability coefficient of the backfill soil in the trench can obviously restrain the floatation of the pipeline. If necessary, the depth of the backfill trench can be deepened in order to further control the floatation of the pipeline.



(a)



(b)

Figure 13. Floating displacement and MAX EPWPR time series for the pipeline, by backfill soil permeability. (a) Time histories of floating displacement; (b) Distributions of the MAX EPWPR.

3.4. Discussion

This paper makes a significant contribution to understanding the behavior of fully buried pipelines under combined wave and current loadings using a poro-elastoplastic approach. It addresses a critical aspect of marine pipeline engineering with a robust theoretical framework and practical insights. It provides valuable references for the construction of practical marine engineering projects.

Typically, the finite deformation is considered in the simulation of sand liquefaction. In this study, small strain elements were used to simulate the liquefaction and deformation of the sandy seabed, resulting in accurate simulation results. A comparative analysis with traditional finite deformation will be conducted subsequently. The used model cannot consider the noncoaxial behavior of the sand. Taking the influence of the noncoaxial property into account in the model is ongoing, and the wave in the real case can be high-order and nonlinear. The liquefaction mechanism of high-order and nonlinear waves is our future work.

4. Conclusions

This study presents a two-dimensional coupled approach with a poro-elastoplastic theory to study the floatation of pipelines with the combined loading of waves and currents. The cyclic constitutive model incorporates the structural, overconsolidation, and anisotropic characteristics of soil, facilitating an accurate portrayal of the static and dynamic behaviors of both sandy and clayey soils, rendering it more comprehensive compared to the elastoplastic model. Based on the data, the following conclusions can be drawn, which provide references for practical engineering design, construction, and risk protection.

(1) Pipeline floatation is broken into two phases. In the initial phase, the pipeline floats slowly under the cyclic loading. In the subsequent phase, when the backfill soil in the middle position of the pipeline begins to liquefy, the floating displacement increases obviously. The pipeline acts as a boundary restriction, strengthening the backfill soil surrounding it and hastening the dissipation of excess pore water pressure. Meanwhile, a hard-to-liquefy area is formed under the seabed pipeline. In the construction of submarine seabed pipeline engineering, particular attention should be paid to the risk of liquefaction of the overlying soil above the seabed pipeline. Measures such as reinforcing the upper soil layer or increasing the permeability coefficient could be adopted to reduce the risk of liquefaction.

(2) The pipeline's floating displacement rises as the current velocity (u), wave period (T), and wave height (H) increase. As current velocity (u), wave period (T), and wave height (H) increase, so too does the liquefaction depth at the pipeline's bottom. In real engineering applications, wave blocking or dissipation methods could be implemented to limit the risk of the seabed pipeline floatation.

(3) The floating displacement of the pipeline and the depth of the liquefaction at the bottom diminish as the permeability of the backfill trench increases. Increasing the permeability of the trench's backfill soil can significantly restrain the floatation of the pipeline.

This study explains the mechanism of fully buried pipeline floatation in practical engineering and analyzes the effects of wave height, wave period, current velocity, and the permeability coefficient of the overlying soil. Based on the conclusions of this study, measures to control the floatation of fully buried pipelines in engineering include increasing the pipeline's weight with concrete coatings or weight blocks, embedding the pipeline in the seabed, securing it with suction or pile anchors, using chains or cables to fix it to anchor points, adjusting buoyancy with control modules or heavy materials, preventing gas accumulation, constructing breakwaters, installing wave energy absorbers, and replacing the surrounding soil with more permeable materials to enhance stability.

Author Contributions: Conceptualization, J.L. and C.L.; Methodology, J.L. and G.Y.; Software, J.L. and G.Y.; Formal analysis, J.L. and C.L.; Investigation, J.L., L.L. and C.L.; Writing—original draft, J.L.

and C.L.; Writing—review & editing, J.L. and C.L.; Supervision, L.L. and G.Y. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: Authors Jian Leng and Libin Liu were employed by the company CCCC Third Harbor Engineering Co., Ltd. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

References

1. Meniconi, S.; Brunone, B.; Tirello, L.; Rubin, A.; Cifrodelli, M.; Capponi, C. Transient tests for checking the Trieste subsea pipeline: Towards the field tests. *J. Mar. Sci. Eng.* **2024**, *12*, 374. [CrossRef]
2. Meniconi, S.; Brunone, B.; Tirello, L.; Rubin, A.; Cifrodelli, M.; Capponi, C. Transient tests for checking the Trieste subsea pipeline: Diving into fault detection. *J. Mar. Sci. Eng.* **2024**, *12*, 391. [CrossRef]
3. Zhang, J.; Wang, Y.; Luo, X. Influence of wave parameters on subsea pipeline stability. *Ocean Eng.* **2017**, *136*, 10–20.
4. Wang, P.; Feng, S.; Liu, X. Combined wave-current effects on pipeline flotation. *Mar. Struct.* **2019**, *65*, 110–123.
5. Liang, W.; Zhang, H. Stability analysis of submarine pipelines under combined wave and current loading. *Ocean Eng.* **2020**, *209*, 107508.
6. Qi, W.; Gao, P. Cyclic loading effects on the stability of subsea pipelines. *J. Offshore Mech. Arct. Eng.* **2014**, *136*, 021303.
7. Liang, W.; Jeng, D.S.; Zhang, H. Numerical study on the interaction between waves, currents, and seabed around subsea pipelines. *Appl. Ocean Res.* **2020**, *94*, 101941.
8. Liu, H.; Chen, Y.; Zhang, L. Numerical simulation of pipeline flotation in liquefied soils. *J. Geotech. Geoenviron. Eng.* **2016**, *142*, 04016013. [CrossRef]
9. Feng, S.; Liu, X.; Zhang, Y. Numerical study of wave-induced seabed response around submarine pipelines. *Mar. Georesour. Geotechnol.* **2018**, *36*, 601–612.
10. Liu, Y.; Ye, J.; Jeng, D.S. Numerical modeling of wave-induced seabed response around marine structures. *J. Waterw. Port Coast. Ocean Eng.* **2018**, *144*, 04018015.
11. Chen, H.; Zhang, J.; Tong, L.; Sun, K.; Guo, Y.; Wei, C. Experimental study of soil responses around a pipeline in a sandy seabed under wave-current load. *Appl. Ocean Res.* **2023**, *130*, 103409. [CrossRef]
12. Sumer, B.M.; Hatipoglu, F.; Fredsøe, J.; Sumer, S.K. The sequence of sediment behavior during wave-induced liquefaction. *Sedimentology* **2006**, *53*, 611–629. [CrossRef]
13. Liang, W.; Yang, K. Two-way coupling model for dynamic seabed response around twin pipelines under combined wave and current loading. *Mar. Georesour. Geotechnol.* **2018**, *36*, 832–846.
14. Zhang, Y.; Ma, G. Experimental study on the response of sandy seabed to regular waves. *Ocean Eng.* **2023**, *150*, 108789.
15. Wang, J.; Liu, Q.; Li, X. Investigation of wave-induced seabed response using physical model tests. *Coast. Eng.* **2023**, *185*, 103809.
16. Smith, A.; Johnson, B. Wave-induced sediment transport and bed morphology changes in laboratory wave tank experiments. *J. Geophys. Res. Ocean.* **2023**, *128*, 982–996.
17. Liu, H.; Shan, H.; Jia, Y. Dynamic Response of Seabed Sediments under Flow Conditions: Insights from Centrifuge Experiments and Numerical Simulation. *J. Ocean Eng.* **2023**, *35*, 245–258.
18. Xu, F.; Mu, X.; Zhen, W. Deformation Characteristics of Seabed Soils under Different Flow Velocities: A Centrifuge Experiment Study. *Mar. Geotech.* **2024**, *28*, 112–125.
19. Zhao, J.; Wang, X.; Zhang, M. Simulation of Seabed Response under Storm Surge and Wave Action using Centrifuge Experiments. *Coast. Eng.* **2024**, *45*, 321–335.
20. Cheng, A.H.D.; Liu, P.L.F. Seepage force on a pipeline buried in a poroelastic seabed under wave loading. *Appl. Ocean Res.* **1986**, *8*, 22–32. [CrossRef]
21. McDougal, W.G.; Davidson, S.H.; Monkmeyer, P.L.; Sollitt, C.K. Wave-induced forces on buried pipeline. *J. Waterw. Port Coast. Ocean. Eng.-ASCE* **1988**, *114*, 220–236. [CrossRef]
22. Zhang, X.L.; Jeng, D.S.; Luan, M.T. Dynamic response of a porous seabed around pipeline under three-dimensional wave loading. *Soil Dyn. Earthq. Eng.* **2011**, *31*, 785–791. [CrossRef]
23. Zhao, H.Y.; Jeng, D.S.; Guo, Z.; Zhang, J.S. Two-dimensional model for pore pressure accumulations in the vicinity of a buried pipeline. *J. Offshore Mech. Arct. Eng.* **2014**, *136*, 042001. [CrossRef]
24. Zhao, H.Y.; Jeng, D.S.; Liao, C.C. Effects of cross-anisotropic soil behavior on the wave-induced residual liquefaction in the vicinity of pipeline buried in elasto-plastic seabed foundations. *Soil Dyn. Earthq. Eng.* **2016**, *80*, 40–55. [CrossRef]

25. Ye, J.H.; He, K.P. Dynamics of a pipeline buried in loosely deposited seabed to nonlinear wave & current. *Ocean Eng.* **2021**, *232*, 109127.
26. Ye, J.H.; Lu, Q.Y. Seismic dynamics of a pipeline shallowly buried in loosely deposited seabed foundation. *Ocean Eng.* **2022**, *243*, 110194. [CrossRef]
27. Zhang, F.; Ye, B.; Noda, T.; Nakano, M.; Nakai, K. Explanation of cyclic mobility of soils: Approach by stress-induced anisotropy. *Soils Found.* **2007**, *47*, 635–648. [CrossRef]
28. Zhang, F.; Ye, B.; Ye, G.L. Unified description of sand behavior. *Front. Archit. Civ. Eng. China* **2011**, *5*, 121–150. [CrossRef]
29. Hsu, H.C.; Chen, Y.Y.; Hsu, J.R.C. Non-linear water waves on uniform current in lagrangian coordinates. *J. Nonlinear Math. Phys.* **2009**, *16*, 47–61. [CrossRef]
30. Ye, J.H.; Jeng, D.S. Response of porous seabed to nature loadings: Waves and currents. *J. Eng. Mech.* **2012**, *138*, 601–613. [CrossRef]
31. Ye, B.; Muramatsu, D.; Ye, G.L.; Zhang, F. Numerical assessment of vibration damping effect of soilbags. *Geosynth. Int.* **2011**, *18*, 159–168. [CrossRef]
32. Hashiguchi, K. Elasto-plastic constitutive laws of granular materials, Constitutive Equations of Soils. In Proceedings of the 9th ICFSME, Session 9, Tokyo, Japan, 10–15 July 1977; pp. 73–82.
33. Asaoka, A.; Nakano, M.; Noda, T. Super loading yield surface concept for the saturated structured soils. In Proceedings of the 4th European Conference on Numerical Methods in Geotechnical Engineering, Udine, Italy, 14–16 October 1998; Volume 98, pp. 232–243.
34. Ye, B.; Ye, G.L.; Zhang, F.; Yashima, A. Experiment and numerical simulation of repeated liquefaction-consolidation of sand. *Soils Found.* **2007**, *47*, 547–558. [CrossRef]
35. Ye, B.; Ye, G.L.; Zhang, F. Numerical modeling of changes in anisotropy during liquefaction using a generalized constitutive model. *Comput. Geotech.* **2012**, *42*, 62–72. [CrossRef]
36. Bao, Y.F.; Ye, G.L.; Ye, B.; Zhang, F. Seismic evaluation of soil-foundation-superstructure system considering geometry and material nonlinearities of both soils and structures. *Soils Found.* **2012**, *52*, 257–278. [CrossRef]
37. Bao, X.H.; Morikawa, Y.; Kondo, Y.; Nakamura, K. Shaking table test on reinforcement effect of partial ground improvement for group-pile foundation and its numerical simulation. *Soils Found.* **2012**, *52*, 1043–1061. [CrossRef]
38. Bao, X.H.; Ye, G.L.; Ye, B.; Sago, Y.; Zhang, F. Seismic performance of SSPQ retaining wall-Centrifuge model tests and numerical evaluation. *Soil Dyn. Earthq. Eng.* **2014**, *61*, 63–82. [CrossRef]
39. Xia, Z.F.; Ye, G.L.; Wang, J.H.; Ye, B.; Zhang, F. Fully coupled numerical analysis of repeated shake-consolidation process of earth embankment on liquefiable foundation. *Soil Dyn. Earthq. Eng.* **2010**, *30*, 1309–1318. [CrossRef]
40. Zienkiewicz, O.C.; Shiomi, T. Dynamic Behavior of Saturated Porous Media: The Generalized Biot Formulation and Its Numerical Solution. *Int. J. Numer. Anal. Methods Geomech.* **1984**, *8*, 71–96. [CrossRef]
41. Oka, F.; Yashima, A.; Shibata, T.; Kato, M.; Uzuoka, R. FEM-FDM coupled liquefaction analysis of a porous soil using an elasto-plastic model. *Appl. Sci. Res.* **1994**, *52*, 209–245. [CrossRef]
42. Jeng, D.S.; Ou, J. 3D models for wave-induced pore pressure near breakwater heads. *Acta Mech.* **2010**, *215*, 85–104. [CrossRef]
43. Jin, Y.; Bao, X.; Kondo, Y.; Zhang, F. Numerical evaluation of group-pile foundation subjected to cyclic horizontal load. *Front. Archit. Civ. Eng. China* **2010**, *4*, 196–207. [CrossRef]
44. Sumer, B.M.; Fredsøe, J.; Christensen, S.; Lind, M.T. Sinking/floatation of pipelines and other objects in liquefied soil under waves. *Coast. Eng.* **1999**, *38*, 53–90. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Numerical Modelling of the Hydrodynamic Performance of Biodegradable Drifting Fish Aggregating Devices in Currents

Tongzheng Zhang ¹, Junbo Zhang ², Qinglian Hou ², Gangyi Yu ², Ao Chen ², Zhiqiang Liu ^{2,*} and Rong Wan ^{1,*}

¹ National Engineering Research Center for Oceanic Fisheries, Key Laboratory of Sustainable Exploitation of Oceanic Fisheries Resources, College of Marine Living Resource Sciences and Management, Shanghai Ocean University, Shanghai 201306, China; zhangtongzheng7@163.com

² College of Marine Living Resource Sciences and Management, Shanghai Ocean University, Shanghai 201306, China

* Correspondence: lzhiqiang716@126.com (Z.L.); rongwan@shou.edu.cn (R.W.)

Abstract: Fish Aggregating Devices (FADs) are essential supplementary structures used in tropical tuna purse-seine fishing. They are strategically placed to attract tuna species and enhance fishing productivity. The hydrodynamic performance of FADs has a direct effect on their structural and environmental safety in the harsh marine environment. Conventional FADs are composed of materials that do not break down naturally, leading to the accumulation of waste in the ocean and potential negative effects on marine ecosystems. Therefore, this work aimed to examine the hydrodynamic performance of biodegradable drifting FADs (Bio-DFADs) in oceanic currents by numerical modelling. The Reynolds-averaged Navier–Stokes equation was used to solve the flow field and discretized based on the realizable k - ϵ turbulence model, employing the finite volume method. A set of Bio-DFADs was developed to assess the hydrodynamic performance under varying current velocities and attack angles, as well as different balsa wood diameters and sinker weights. The results indicated that the relative current velocity significantly affected the relative velocity of Bio-DFADs. The relative length of the raft significantly affected both the relative velocity and the relative wetted area in a pure stream. Finally, the diameter of the balsa wood affected the drift velocity, and the sinker's relative weight affected the hydrodynamic performance of the Bio-DFADs.

Keywords: biodegradable drifting Fish Aggregating Devices (Bio-DFADs); hydrodynamic performance; fluid–structure interaction; current

1. Introduction

Approximately 66% of the worldwide tuna harvest is obtained by tuna purse-seine vessels, with 36% originating from sets of drifting Fish Aggregating Devices (DFADs) in tropical tuna fisheries [1]. Tropical tuna purse-seine vessels operate in the equatorial waters of the pelagic oceans. They use a large number of DFADs to gather tuna species and improve their fishing efficiency. It is estimated that over 100,000 DFADs are deployed annually by these vessels. Numerous studies have examined this practice [2–9]. However, the majority of traditional DFADs consist of synthetic and non-biodegradable netting materials, which contribute to marine litter, the entanglement of sensitive species, and other negative impacts on the ecosystem. These consequences have been documented in various studies [10–17]. Most DFADs endure a lengthy amount duration underwater in the ocean and are subjected to a range of oceanographic conditions, such as currents, waves, and biofouling. The proliferation of DFADs in the open ocean is an issue, considering their adverse effects on the marine environment. Research on the hydrodynamic performance of DFADs helps elucidate the structural features that are well-suited for maritime environments. Additionally, different materials possess distinct fracture strengths that significantly influence the safety of structures. The hydrodynamic performance of the Bio-DFADs in

oceanic currents is also crucial for identifying the overall drifting characteristics, functioning as free-drifting marine infrastructure. Consequently, it is essential to focus on the hydrodynamic performance and material concerns in practical conditions to successfully apply DFADs.

Extensive studies have been conducted to examine the numerical methodology for analyzing stiff and flexible bodies [18–25]. In recent decades, several investigators have studied the hydrodynamic performance of DFADs. Investigating the hydrodynamic performance of DFADs is necessary to improve the safety and efficiency of fish aggregation [26,27]. To enhance the environmental safety of DFADs in marine environments, a variety of DFADs have been constructed using biodegradable materials. These materials were employed to investigate the impact of various structural characteristics on wave conditions through experimental tests and numerical simulations [18,26]. In addition to waves, the current is an essential factor in the design of marine infrastructure. Lee et al. (2018) [28] deployed the particle image velocimetry method to analyze the velocity and turbulence intensity of high-risk net-type and low-risk rope-type DFADs in multiple current conditions. The results showed a notable difference in the drift velocity of the low-risk rope-type DFADs.

However, prior research has concentrated on the motion responses of DFADs in marine waves. The hydrodynamic characteristics of the Bio-DFADs in the presence of a current have not been fully investigated. Notably, Bio-DFADs comprise a combination of a small-scale raft and a flexible structure. The flexible structure of the Bio-DFADs, especially when submerged, is often exposed to fluid currents. Moreover, biofouling has an impact on the hydrodynamic properties of biodegradable rafts and ropes, which impairs their structural integrity. Determining the drift velocity of Bio-DFADs in various currents aids in clarifying the optimal deployment strategy. Hence, it is necessary to investigate the hydrodynamic properties of Bio-DFADs in flowing currents.

This study presents a numerical modelling scheme that evaluates the interaction between current and Bio-DFADs in three dimensions. The Bio-DFADs contained biodegradable materials, such as bamboo and cotton rope, making them environmentally acceptable for the marine ecosystem. Herein, STAR-CCM+ software 14.02 was used to compute the hydrodynamic performance of the Bio-DFADs under various current conditions. We adopted the finite volume method to discretize the Reynold-averaged Naviers–Stokes equations (RANS). The Euler overlay method was used to reduce the level of reflection at the boundaries. Finally, the motion responses of the Bio-DFADs in current were derived by employing the dynamic fluid–body interaction (DFBI) model.

2. Materials and Methods

2.1. Description of the Bio-DFADs

This study aimed to perform numerical simulations of Bio-DFADs in the presence of steady currents. Figure 1 depicts a schematic representation of the Bio-DFADs, and Table 1 lists the major properties of the Bio-DFADs. The Bio-DFAD model comprises three components: raft, rope, and sinker. The raft is composed of two perpendicular layers of equidistant balsa wood sticks. The balsa wood raft eventually degrades, which is beneficial when the Bio-DFADs are lost or abandoned. This type of raft has been employed for constructing the Bio-DFADs in the actual ocean [16]. A total of five buoys, which are agreed to be used in the design of DFADs, are attached to the top part of the raft. The buoys, made of Ethylene Vinyl Acetate Copolymer (EVA) material, provide buoyancy to the overall structure. Each float is shaped like an ellipsoidal sphere, with a length of 24 cm and both width and height of 12 cm. The rope is fastened to the bottom section of the raft at one end, and on the other end, it is linked to a steel sinker weighing 2.00, 4.00, 6.00, 8.00, 10.00, 12.00, and 14.00 kg.

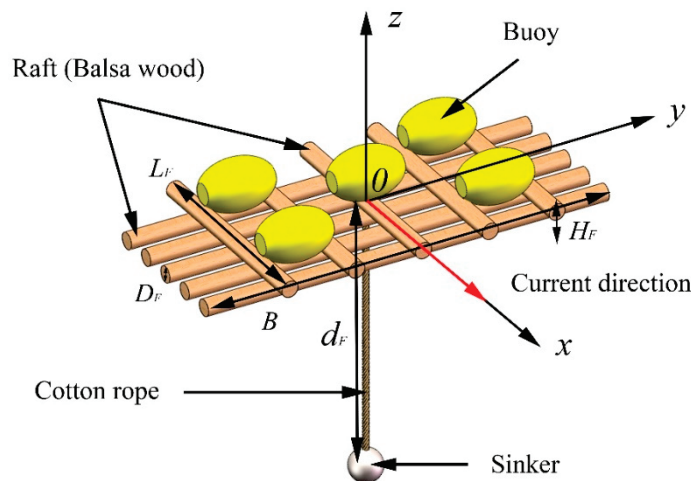


Figure 1. A schematic of the Bio-DFAD model.

Table 1. The main characteristics of the Bio-DFAD model.

Main Characteristics	Parameter	Value
Length of the raft	L_F	0.50, 1.00, 1.50 m
Width of the raft	B	1.00 m
Height of the raft	H_F	0.06, 0.08, 0.10, 0.12, 0.14, 0.16 m
Diameter of the balsa wood stick	D_F	0.03, 0.04, 0.05, 0.06, 0.07, 0.08 m
Length of the rope	d_F	0.60 m
Weight of the sinker	W_{Sinker}	2.00, 4.00, 6.00, 8.00, 10.00, 12.00, 14.00 kg

2.2. Test Conditions

The Bio-DFADs were subjected to heave, pitch, and surge motions in pure currents during the numerical simulations. We investigated a total of ten current speed cases based on monitoring data from the tuna purse-seine fleets. The current velocity, U , was specified as 0.10, 0.20, 0.30, 0.40, 0.50, 0.60, 0.70, 0.80, 0.90, and 1.00 m/s. Various attack angles, denoted as α , were also examined, as depicted in Figure 2. The test conditions are presented in Table 2. The computational fluid dynamics (CFD) simulations of Bio-DFADs in currents were initially conducted in tranquil water circumstances, followed by the consideration of 10 different current regimes. The 10 current speeds were derived based on the concept of a flat wave in volume of fluid (VOF) wave models.

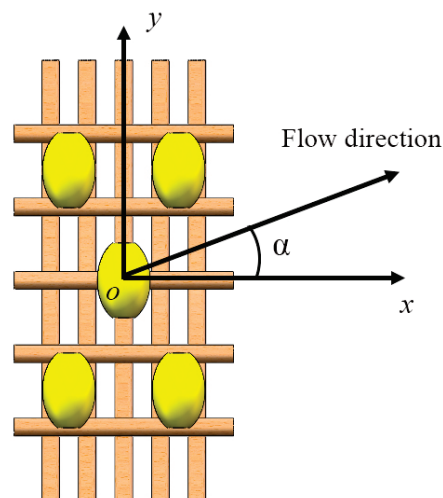


Figure 2. The attack angle between the Bio-DFADs and flow direction.

Table 2. Test conditions of the Bio-DFAD simulations.

Cases	Current Velocity U (m/s)	Attack Angle α (°)
Calm water	0.10, 0.20, 0.30, 0.40, 0.50, 0.60, 0.70, 0.80, 0.90, 1.00	0, 90

2.3. Numerical Setup

2.3.1. Governing Equations and Turbulence Model

Reynolds-averaged Navier–Stokes (RANS) simulations using the realizable k - ε turbulence model with two-layer all y + wall treatment are adopted in the present study [29]. The realizable k - ε model is considered to be more effective than other turbulence models in accurately estimating the distribution of the dissipation rate [30]. The RANS method is a feasible and widely acknowledged solution for turbulence modelling and can provide a rational prediction for turbulent flows from ocean structures. The mass and momentum conservation equations are solved using CFD software (STAR-CCM+) and the overset mesh method. For incompressible flows, the averaged continuity and momentum equations can be expressed as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \rho g_i + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \overline{u'_i u'_j} \right] \quad (2)$$

where t represents the time, ρ represents the fluid density, P represents the mean pressure, S represents the closed surface, U_i represents the average velocity ($i, j = 1, 2, 3$ represent the x, y , and z directions in the Cartesian coordinate system), g_i represents the gravitational acceleration, μ represent the dynamic viscosity of the fluid, u_i , and u_j denote the mean velocities in directions i and j , respectively, u'_i and u'_j denote the turbulence velocities in directions i and j , respectively, and $-\overline{u'_i u'_j}$ denote the Reynolds stress.

Typically, the RANS equations are solved using the finite volume method. The governing equations are discretized using second-order schemes in space and second-order implicit schemes in time. The VOF approach is used to accurately distinguish between the two phases (i.e., air and water phases) of the free surface (Hirt and Nichols, 1981) [31]. The convective term in the VOF transport equations is discretized using the high-resolution interface capturing approach proposed by Hirt and Nichols in 1981 [31]. The turbulence modelling is based on a realizable k - ε model with a two-layer all y + wall treatment. The equations for the transmission of turbulent kinetic energy, k , and the specific dissipation rate of turbulence, ε , are the following:

k equation

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \quad (3)$$

ε equation

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (4)$$

where G_k represents the turbulent kinetic energy generated by the mean velocity gradient, σ_k and σ_ε denote constants representing the turbulent Prandtl numbers, and C_μ , $C_{1\varepsilon}$, and $C_{2\varepsilon}$ denote constants. The model constants for the realizable k - ε turbulence model denote $C_\mu = 0.09$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $\sigma_k = 1.0$, and $\sigma_\varepsilon = 1.3$ [32,33].

2.3.2. Computational Domain and Boundary Conditions

Figure 3 depicts the computational domain and boundary conditions of the scaled Bio-DFAD model, which was constructed using the Cartesian coordinate system. The coordinate system is center at a position that is $4 L_F$ from the inlet boundary and $10 L_F$ from the outlet boundary. For the width of the computational domain, to reduce the computational cost, the symmetry boundary is set at the place of half width, $3 B$ from the side boundary. The water depth, denoted by h , is 0.80 m. The coordinate system of the initial position of the Bio-DFAD aligns with the computational domain. Two overlapping sections were used in the computational domain. The overset method is employed to capture the motion responses of the Bio-DFADs. The velocity inlet condition was imposed on the inlet, top, and bottom limits of the domain, with the stated velocity of the flat VOF wave. A pressure outlet condition was imposed on the outlet boundary of the domain, specifying the pressure as the hydrostatic pressure of the flat VOF wave. In addition, the front and rear limits were established as symmetry conditions. To decrease the computational workload, just half of the Bio-DFADs model was used, with the surfaces designated as non-slip wall conditions.

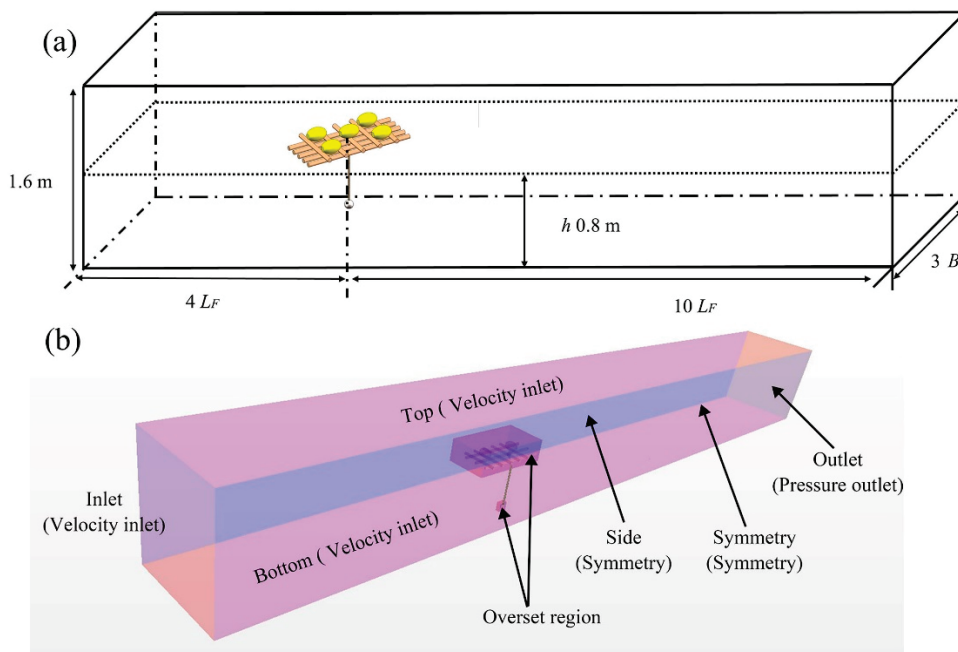


Figure 3. Computational domain and boundary conditions with the scaled Bio-DFADs model: (a) the global 3D computational domain; (b) boundary conditions.

This research primarily focuses on numerical simulations of the Bio-DFAD's hydrodynamic performance. Owing to its low cost and short duration, CFD simulations are extensively employed in researching the hydrodynamic performance of oceanic structures. The CFD numerical software STAR-CCM+ is used to examine the hydrodynamic properties of the Bio-DFADs and investigate their drift velocity under varying current velocities. The VOF method and six degrees of freedom (6-DOF) solver were used to analyze the hydrodynamic performance of the Bio-DFADs, specifically, the surge, pitch, and heave motions of the Bio-DFADs. The 6-DOF solver modified the position and orientation of the DFAD body. The DFBI model was used to simulate the movements of the Bio-DFADs in reaction to pressure and shear forces inside the flow domain, as well as the force exerted by the catenary. In the DFBI model, the RANS solver was used to compute the dynamic forces and moments exerted on the Bio-DFADs and adjust the position of the rigid body. The rope in the Bio-DFADs system was represented by a catenary in the body couplings of the STAR-CCM+ package, as illustrated in Figure 3b. The catenary coupling model was used to simulate the elastic, quasi-stationary rope that hangs between two endpoints, namely the

raft and sinker, and is affected by its own weight in the gravitational field. The parametric equations that describe the shape of the catenary in the Cartesian coordinate system of the computational domain are expressed as follows (STAR-CCM+, 2019) [34]:

$$x = au + b \sin(u) + \alpha \quad (5)$$

$$y = a \cosh(u) + \frac{b}{2} \sinh^2(u) + \beta \quad (6)$$

$$\text{for } u_1 \leq u \leq u_2 \quad (7)$$

$$a = \frac{c}{\lambda_0 g} \quad (8)$$

$$b = \frac{ca}{DL_{eq}} \quad (9)$$

$$c = \frac{\lambda_0 L_{eq} g}{\sinh(u_2) - \sinh(u_1)} \quad (10)$$

where g is the value of gravitational acceleration, D represents the stiffness of the catenary, and λ_0 and L_{eq} represent the mass per unit length and relaxation length of the catenary, respectively. The integration constants α and β depend on the position of the two endpoints and the total mass of the catenary. The positions of the two endpoints in the parameter space of the catenary are denoted by u_1 and u_2 . The curve parameter, denoted as u , is directly linked to the inclination angle Φ of the catenary curve:

$$\tan \Phi = \sinh(u) \quad (11)$$

To enhance the efficiency of the simulations and reduce computational time, the CFD simulation employs the VOF wave-damping approach, as proposed by Choi and Yoon in 2009 [35]. In addition, the VOF wave-damping function was implemented at the outlet border to attenuate the vertical disturbances that propagate towards it and reduce the wave reflections at the boundaries. The wave-damping zone at the outlet side of the computational domain has a length of $2 L_F$. An additional factor representing resistance is incorporated into the momentum equation for the w -velocity, as described by Gu et al. (2018) [36].

$$S_z^d = \rho(f_1 + f_2|w|) \frac{e^k - 1}{e^1 - 1} w \quad (12)$$

$$k = \left(\frac{x - x_{sd}}{x_{ed} - x_{sd}} \right)^{n_d} \quad (13)$$

where ω represents the vertical velocity component, and f_1, f_2 , and n_d are parameters of the damping model. Furthermore, x_{sd} and x_{ed} indicate the start and end locations of the wave-damping zone, respectively.

2.3.3. Motion Equation of the Bio-DFADs

The Bio-DFADs in this study exhibit unrestricted movement in the current, including heave, pitch, and surge motions. The equations of motion for the Bio-DFADs can be represented as the following:

Translation equation

$$m \frac{dv}{dt} = f \quad (14)$$

Rotation equation

$$M \frac{d\omega}{dt} + \omega \times M\omega = n \quad (15)$$

where m represents the mass of the body, f represents the resultant force acting on the body, and v represents the velocity of the centre of mass. The tensor M represents the moment of inertia, ω represents the angular velocity of the rigid body, and n represents the resultant moment acting on the body, which encompasses the pressure moment, shear moment, and external moment induced by the catenary.

The tensor of the moments of inertia, moment and resultant force acting on the body can be written as:

$$M = \begin{pmatrix} M_{xx} & M_{xy} & M_{xz} \\ M_{xy} & M_{yy} & M_{yz} \\ M_{xz} & M_{yz} & M_{zz} \end{pmatrix} \quad (16)$$

$$n = f_r(n_p + n_\tau + \sum n_{ext}) \quad (17)$$

$$f = f_r(F_p + F_\tau + F_g + \sum F_{ext}) \quad (18)$$

The tensor M is symmetric, it can be defined by two vectors: one specifying the principal components, (M_{xx}, M_{yy}, M_{zz}) along the diagonal, and another specifying the off-diagonal components, (M_{xy}, M_{xz}, M_{yz}) . F_{ext} and n_{ext} represent the extra forces and moments, F_g is the gravity force, F_p and n_p are the fluid pressure force and moment acting on the body, F_τ and n_τ are the fluid shear force and moment acting on the body, and f_r is the time-ramping function. In this study, there are no extra force and moments forcing the model to move.

2.3.4. Mesh Resolution

The trimmed cell mesher, prism layer mesher, and surface remesher were used to discretize the computational domain. These meshing techniques are well-suited for the intricate geometric characteristics of the Bio-DFAD system. Given that the primary focus of the numerical simulation was the surfaces of the Bio-DFAD model and the surrounding regions, special treatment was applied to the meshes on these surfaces. Ultimately, the appropriate mesh size was determined using a mesh optimization test to ensure the stability and convergence of the simulation. The objective indices chosen for the test were the drift velocity (V), time step, and maximum mesh size. Velocity monitoring at various distances ($D = 1.00, 2.00$, and 3.00 m) from the inflow border (with a current velocity of $U = 0.10$ m/s) was undertaken individually. The calculation achieved stability when the time step and maximum size in the computational domain were set to 0.005 s and 0.64 m, respectively, as demonstrated in Figure 4.

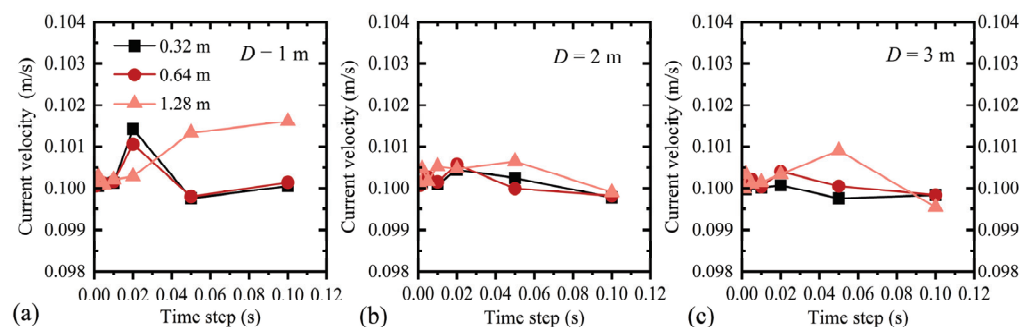


Figure 4. Mesh optimization test with various distances from the inflow border. (a) $D = 1.00$ m, (b) $D = 2.00$ m, (c) $D = 3.00$ m.

Figure 5 shows the current velocity values between numeric and theory calculations. One of the current velocities, 0.50 m/s was selected as the test condition. The monitor point was set at the 1.00 m away from the inlet boundary. Based on the numeric model, the error rate is 0.72% . The results demonstrate that the numerical model established in this study has a high degree of accuracy.

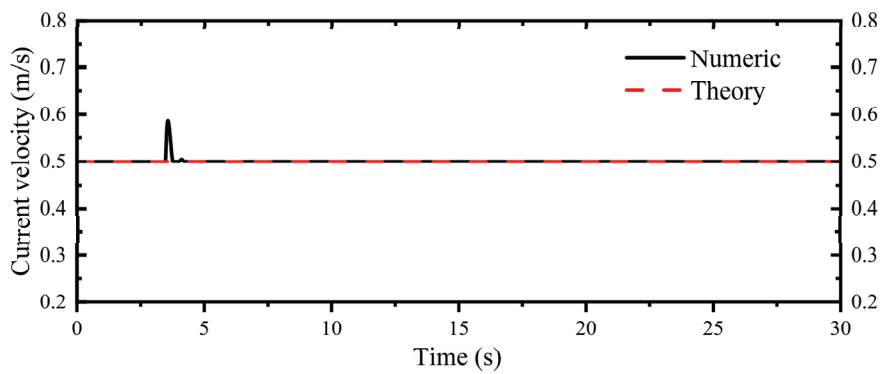


Figure 5. The comparison of current velocity between numeric and theory (0.50 m/s).

Figure 6 depicts the mesh generated in the computational domain. To enhance the precision of the Bio-DFAD responses in currents, two overset mesh zones are used for the raft and sinker. Figure 6 reveals that the meshes in close proximity to the water's surface are more detailed, and the overset meshes near the Bio-DFAD are precisely suited to the body. This allows for accurate simulation of the Bio-DFAD motion responses. To analyze the fluid dynamics surrounding the Bio-DFADs, the thickness of the prism layer is defined as 0.01 m, consisting of five layers. Additionally, a prism layer stretching factor of 1.2 is applied to the wall limits of the Bio-DFADs. The y^+ values vary from 30 to 100. The trimmer mesh is employed to incorporate the Bio-DFAD characteristics and discretize the remaining portion of the computational domain. The mesh size scale, represented by the $\Delta x/\Delta z$ ratio, varies between 4 and 1. Specifically, it is equal to 1 in the two overset mesh sections and 4 in the water-free surface. The time-step interval is defined as 0.005 s. To mitigate the numerical dispersion error [32], the Courant number of the VOF model is adjusted to be less than 0.3. Moreover, mesh refinements are employed to accurately capture the surface waves and decrease the number of cells in the computational domain.

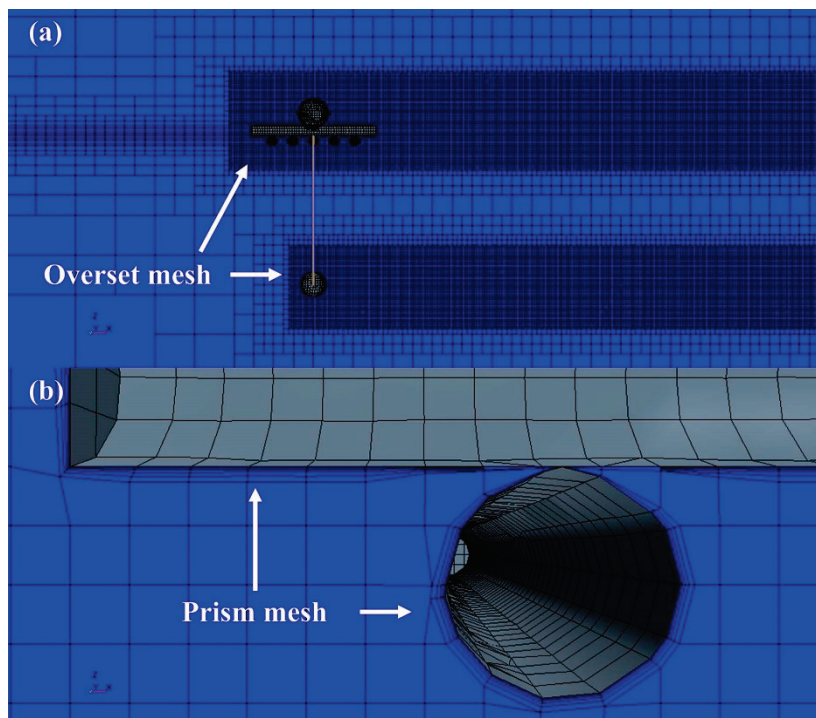


Figure 6. Mesh generation in the computational domain. (a) Overset mesh, (b) Prism mesh.

3. Results and Discussions

3.1. Effect of the Relative Current Velocity on the Bio-DFADs

Figures 7–9 show the volume of fraction, free surface elevation, and velocity contours of the Bio-DFADs at $U = 0.10$ m/s, including the top view, side view, and drift velocity contours at different times when the Bio-DFAD motion became relatively stable under a fixed current velocity.

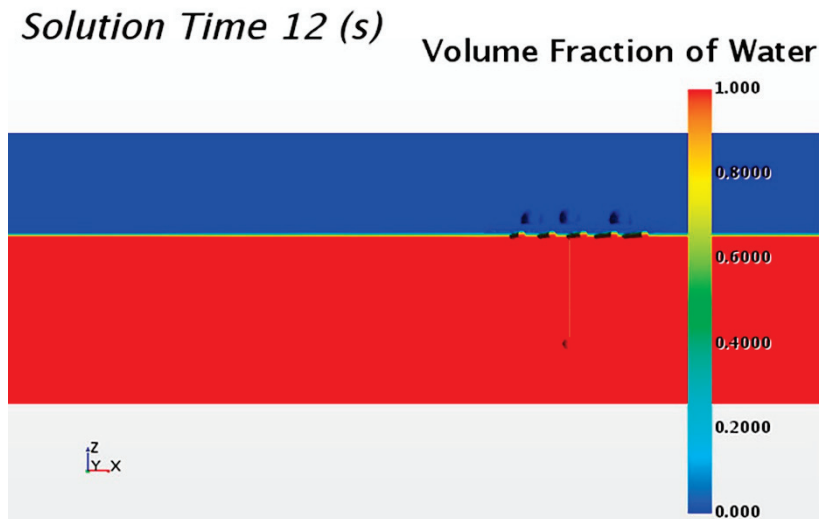


Figure 7. Front view of the volume fraction of water contours of the Bio-DFADs ($U = 0.10$ m/s).

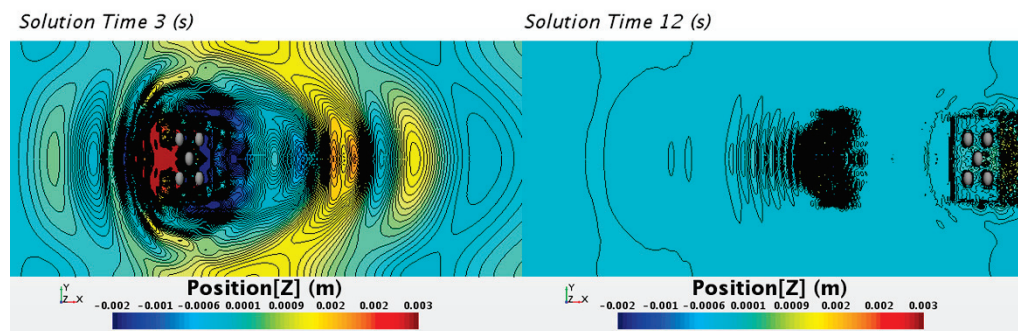


Figure 8. Top view of the free surface elevation contours of the Bio-DFADs ($U = 0.10$ m/s).

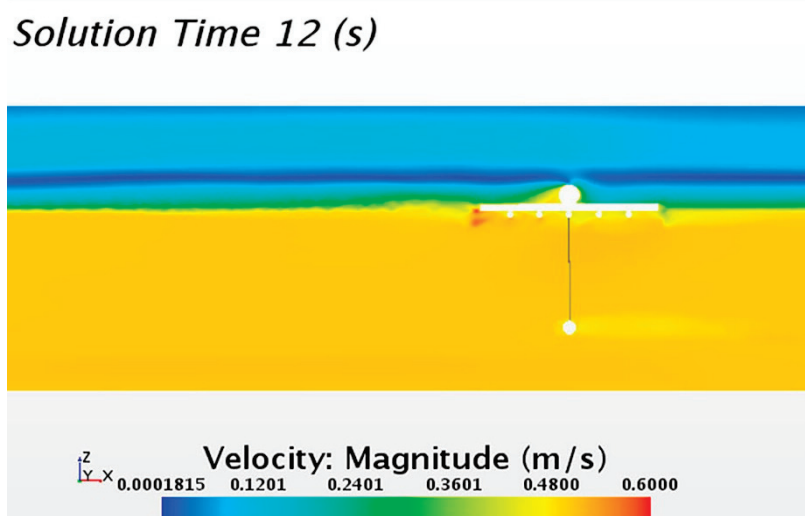


Figure 9. Velocity contours of the Bio-DFADs ($U = 0.10$ m/s).

To test whether current velocity shows generally distinct activity in response to different hydrodynamic performances, the effects of different current velocities (U) on the drift velocity (V) and the effects of relative current velocities (U/B) on relative velocity (V/U) of the Bio-DFADs were investigated (Figure 10), for $D_F = 40$ mm, $W_{\text{Sink}} = 2.00$ kg, and $\alpha = 0^\circ$. Bio-DFADs with three raft lengths (0.50, 1.00, and 1.50 m) were employed in this study. The drift velocity is calculated through the distance drift per unit time ($\Delta d / \Delta t$) in the current direction. The wetted area is calculated by the area covered by the water to the total area of the raft ($S_{\text{Wetted}} / S_{\text{Total}}$), and to indicate the invisibility of the FADs, which is often broken and stolen by other fleets of fishers. As depicted in Figure 10a, the Bio-DFADs exhibited a substantial rise in drift velocity, V , when the current velocity, U , increased. The drift velocity of the model with a balsa wood length of 0.50 m was lower than that of the other two models when the current velocity value was lower. Nevertheless, the disparity in drift velocity among the three models diminished as the current velocity increased. Furthermore, when the current velocity exceeded 0.90 m/s, the drift velocity of the model with a balsa wood length of 0.50 m became slightly greater than the drift velocities of the other two models. Figure 10b depicts the correlation between the relative current velocity, U/B , and the relative velocity, V/U . Initially, the relative velocity of the model with a length of 0.50 m was considerably lower than that of the other two models. The relative velocity had a direct correlation with the balsa wood length, gradually increasing until reaching saturation. The disparity between the 0.50 and 1.00 m models was greater than that between the 1.50 and 1.00 m models. Nevertheless, the disparity diminished as the relative current velocity increased. When the relative current velocity exceeded 0.9, the model with a length of 0.50 m exhibited a somewhat greater relative velocity compared with the other two models. The findings indicated that the relative current velocity had a substantial impact on the relative velocity, and the disparity in relative velocity across the three models tended to be low. The length of the balsa wood also had a significant impact on the relative velocity, although this influence diminished as the relative current velocity increased.

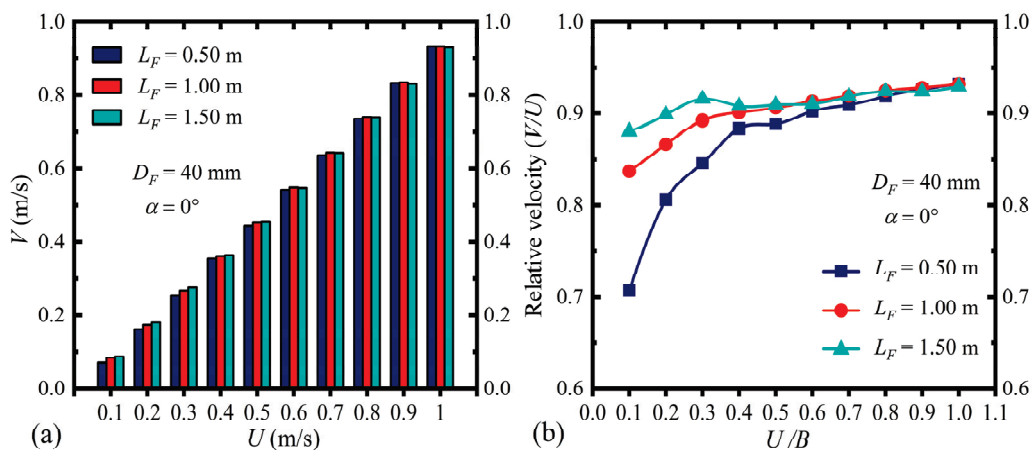


Figure 10. The (a) effects of different current velocities (U) on the drift velocity (V) and (b) effects of relative current velocities (U/B) on the relative velocity (V/U) of the Bio-DFADs.

The proportion of the wetted area ($S_{\text{Wetted}} / S_{\text{Total}}$) for the three models remained stable as the relative current velocity increased. The disparity between the 0.50 and 1.00 m models was greater than that between the 1.00 and 1.50 m models. The wetted area decreased with the increasing length, and the effect diminished as the length increased. A higher wetted area relative to the surroundings suggests a higher level of invisibility. The rope tension slightly increased as the current velocity increased. Additionally, the tension increased as the balsa wood length increased. The findings demonstrated that the relative current velocity had little impact on the relative wetted area and rope tension, but the length affected both the relative wetted area and the relative rope tension.

3.2. Effect of the Relative Length and Attack Angle on the Bio-DFADs

As previously mentioned, the length of Bio-DFADs had a significant effect on relative wetted area and rope tension. The relationships for relative velocity (V/U), relative wetted area (S_{Wetted}/S_{Total}), and relative rope tension ($F/W_{Sinkers}$) with different relative lengths (0.50/1.00, 1.00/1.00, 1.50/1.00, 1.00/0.50, 1.00/1.00, and 1.00/1.50) and attack angles (0° , 90°) are shown in Figure 11. As seen in Figure 11, changes in the relative length and attack angle resulted in changes to the hydrodynamic performance of Bio-DFADs, for $D_F = 40$ mm, $W_{Sinkers} = 2.00$ kg, and $U = 0.50$ m/s.

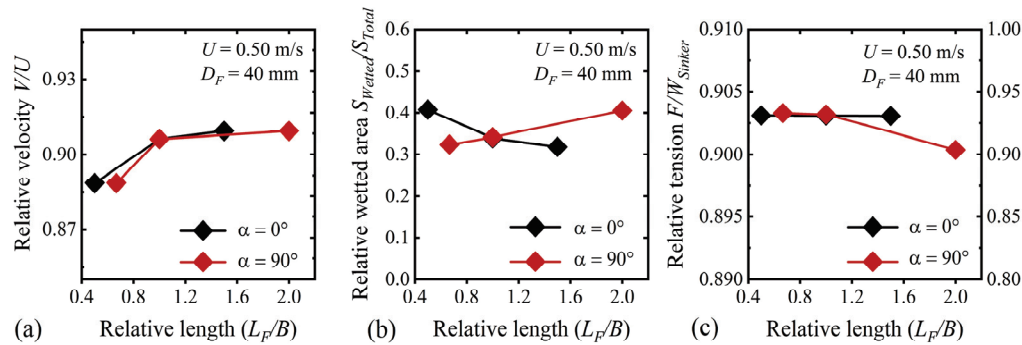


Figure 11. The change in (a) relative velocity (V/B), (b) wetted area (S_{Wetted}/S_{Total}), and (c) rope tension ($F/W_{Sinkers}$) of the Bio-DFADs for different relative lengths and attack angles.

Figure 11a shows the effects of the relative length and attack angle on the relative velocity. The relative velocity was minimized for $\alpha = 0^\circ$, compared with $\alpha = 90^\circ$. When $\alpha = 0^\circ$, the relative velocity increased and then decreased as the relative length increased. When $\alpha = 90^\circ$, the relative velocity increased as the relative length increased. The values at points three and four indeed do not show significant differences. These two points represent the results at different attack angles for a model with an aspect ratio of 1:1. Since most of the raft is submerged in the water, the pressure and drag forces experienced by the model do not differ significantly, leading to negligible differences in relative velocity. As shown in Figure 11b, for $\alpha = 90^\circ$, the relative wetted area was significantly larger than that when $\alpha = 0^\circ$. Additionally, for $\alpha = 0^\circ$, the relative wetted area decreased and then increased as the relative length increased, and the opposite trend was observed for $\alpha = 90^\circ$. As depicted in Figure 11c, as the relative length increased, the relative tension did not change. The relative tension was slightly less for $\alpha = 90^\circ$ than for $\alpha = 0^\circ$. The results indicate that relative length had a significant effect on the relative velocity and relative wetted area under pure current but had almost no influence on the relative tension. Moreover, the attack angle had almost no influence on the hydrodynamic performance of the model. The relative velocity was directly correlated with the relative wetted area.

3.3. Effect of the Relative Diameter of the Balsa Wood on the Bio-DFADs

To examine whether the diameter of the balsa wood affected the hydrodynamic performance of the Bio-DFADs, six diameters of the balsa wood (0.03, 0.04, 0.05, 0.06, 0.07, and 0.08 m) were selected and applied, for $\alpha = 0^\circ$, $L_F = 0.50$ m, and $U = 0.50$ m/s, as depicted in Figure 12.

Figure 12a illustrates the relationship between the relative velocity, V/U , and the relative diameter, D_F/B , of the Bio-DFADs. The enlargement of the diameter initially led to a significant decrease in relative velocity, with a diminishing inclination to drop as the diameter increased. According to Figure 12b, the ratio of the wetted area to the total area, S_{Wetted}/S_{Total} , decreased considerably and subsequently reached a stable state as the diameter increased. Figure 12c illustrates the correlation between the relative diameter and the relative tension, $F/W_{Sinkers}$. As the diameter of the object increased, the tension slightly decreased. As the relative diameter increase, the buoyancy of the Bio-DFADs became greater than the gravity, then the more surface of the model out the water, and lead to the

decrease of the relative wetted area. Due to the decrease of the relative wetted area, the fluid force acting on the model decreases, and then leads to a decline of the relative velocity. The findings indicated that the size of the balsa wood had a notable impact on the velocity and the surface area in contact with water while having minimal effect on the rope tension. The correlation between the relative velocity and the wetted surface area agrees with the results presented in the previous section.

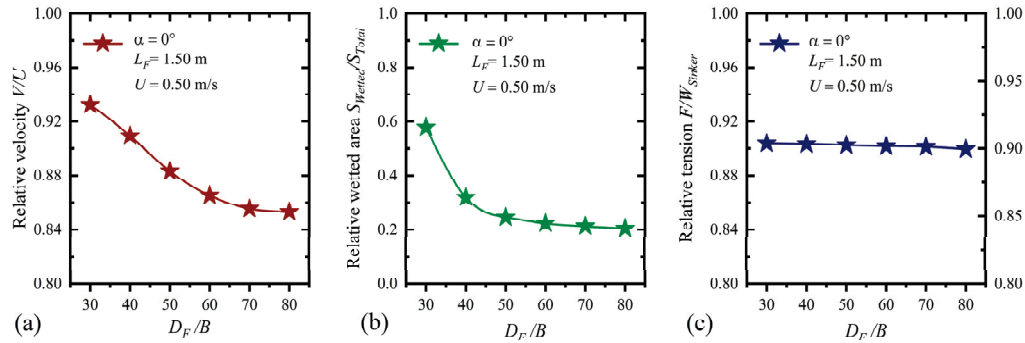


Figure 12. The change in (a) relative velocity (V/B), (b) wetted area (S_{Wetted}/S_{Total}), and (c) rope tension (F/W_{Sinker}) of the Bio-DFADs for different balsa wood diameters.

3.4. Effect of the Relative Weight of the Sinker on the Bio-DFADs

To investigate whether the relative weight of the sinker affected the hydrodynamic performance of the Bio-DFADs, seven sinker weights (2.00, 4.00, 6.00, 8.00, 10.00, 12.00, and 14.00 kg) were selected and applied, for $\alpha = 0^\circ$, $L_F = 0.50$ m, and $U = 0.50$ m/s, as depicted in Figure 13.

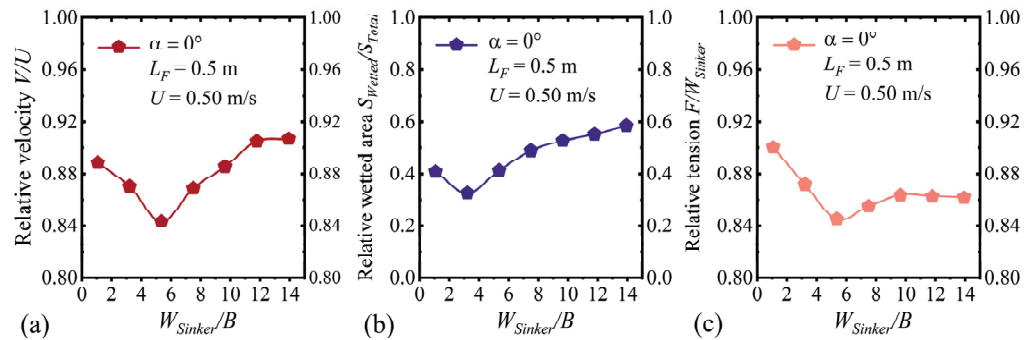


Figure 13. The change in (a) relative velocity (V/U), (b) wetted area (S_{Wetted}/S_{Total}), and (c) rope tension (F/W_{Sinker}) of the Bio-DFADs for different sinker weights.

Figure 13a demonstrates the relationship between the relative velocity, V/U , and the relative weight of the sinker, W_{Sinker}/B . The relative velocity initially decreased and then increased as the relative weight increased. The relative velocity reached a minimum when W_{Sinker}/B was 6. According to Figure 13b, the wetted area also decreased and then increased as the relative weight increased, and the wetted area reached a minimum when W_{Sinker}/B was 4. According to Figure 13c, the relative tension followed a similar trend, initially decreasing and then increasing as the relative weight increased, and the relative tension reached its lowest point when W_{Sinker}/B was 6. Therefore, the weight of the sinker had a notable impact on the hydrodynamic performance of the Bio-DFADs. As the relative weight increase, a minimum value occurs in the relative velocity, relative wetted area, and relative tension, but at different relative weight value. The phenomenon can be explained by the fact that as the weight of the sinker increases, due to the gravity being greater than the fluid force acting on the sinker, the angle between the sinker and rope decreases, then leads to more surface of the raft out the water. Hence, the wetted area declines and the fluid force acting on the Bio-DFADs decreases and leads to the decrease of relative velocity. However, as the weight continues to rise, the fluid force becomes great than the gravity

of sinker, then the angle between the sinker and rope increases, and leads to the relative velocity, relative wetted area, and relative tension increasing. The Bio-DFADs can remain in the fishing area for a predetermined duration while maintaining a reduced speed, but notably, the relative velocity only experiences a drop within a specific range of W_{Sinker}/B .

4. Conclusions

This study investigates the hydrodynamic performance of Bio-DFADs in a pure current using CFD simulations. The effects of the relative current velocity, attack angle, relative diameter of the balsa wood, and relative weight of the sinker on the relative velocity, wetted surface, and rope tension were examined. The key conclusions can be summarized as follows:

- (1) The relative current velocity had a substantial impact on the relative velocity. However, the differences in relative velocity among the three models was modest. Additionally, the relative current velocity had a negligible influence on the relative wetted area and the rope tension.
- (2) The relative length exerted a substantial impact on the relative velocity and the relative wetted area in pure current while having a minimal influence on the relative tension. The hydrodynamic performance of the model was minimally affected by the attack angle. There was a direct correlation between the relative velocity and the relative wetted area.
- (3) The diameter of the balsa wood had a notable impact on the velocity and wetted area but had a minimal effect on the rope tension.
- (4) The sinker's relative weight had a notable impact on the hydrodynamic performance of the Bio-DFADs. The Bio-DFADs can remain in the fishing area for a predetermined duration while moving at a reduced speed. However, the relative velocity was only diminished within a specific range of relative weight of sinker.

This study investigates the hydrodynamic characteristics of the Bio-DFADs, aiming to explore structural forms that combine low drift velocities with biodegradability. The goal is to maximize the retention of fish aggregating devices within the fishing grounds during operations of tuna purse-seine fleets. Additionally, even in cases of damage or sinking, the study aims to minimize marine environmental pollution by employing biodegradable materials.

The present investigation focused on the hydrodynamic performance in undiluted current conditions. Further investigation is required to examine the impact of waves on the hydrodynamic characteristics of various Bio-DFAD geometries. The wave mainly influences the motion responses of the raft and the rope tension. Hence, the wave may have an important influence on the relative velocity, relative rope tension, and relative wetted area. Furthermore, it is necessary to investigate the impact of wave-current interactions on the Bio-DFADs in future research.

Author Contributions: Conceptualization, T.Z., Z.L. and R.W.; methodology, software, validation, formal analysis, investigation, visualization, data curation, T.Z., J.Z., Q.H., G.Y. and A.C.; writing—original draft preparation, T.Z., J.Z. and R.W.; writing—review and editing, T.Z., Z.L. and R.W.; supervision, T.Z., Z.L. and R.W.; project administration, T.Z. and R.W.; funding acquisition, T.Z. and R.W. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by National key R&D Program of China (Project no. 2023YFD2401301); National key R&D Program of China (Project no. 2023YFD2401305); Open Funding Project of the Key Laboratory of Sustainable Exploitation of Oceanic Fisheries Resources, Ministry of Education (Project no. A1-2006-23-200209).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are contained within the article and are available from the corresponding author upon reasonable request.

Acknowledgments: The authors would like to thank the Shanghai Ocean University members for their help with modeling and analysis.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. ISSF. *Status of the World Fisheries for Tuna*; ISSF Technical Report 2022-04; International Seafood Sustainability Foundation: Washington, DC, USA, 2022.
2. Maufroy, A.; Chassot, E.; Joo, R.; Kaplan, D.M. Large-scale examination of spatio-temporal patterns of drifting fish aggregating devices (dFADs) from tropical tuna fisheries of the Indian and Atlantic Oceans. *PLoS ONE* **2015**, *10*, e0128023. [CrossRef] [PubMed]
3. Imzilen, T.; Lett, C.; Chassot, E.; Kaplan, D.M. Spatial management can significantly reduce dFAD beachings in Indian and Atlantic Ocean tropical tuna purse seine fisheries. *Biol. Conserv.* **2021**, *254*, 108939. [CrossRef]
4. Escalle, L.; Hare, S.R.; Vidal, T.; Brownjohn, M.; Hamer, P.; Pilling, G. Quantifying drifting Fish Aggregating Device use by the world's largest tuna fishery. *ICES J. Mar. Sci.* **2021**, *78*, 2432–2447. [CrossRef]
5. Lopez, J.; Roman, M.H.; Lennert-Cody, C.; Maunder, M.N.; Vogel, N. Floating object fishery indicators: A 2020 report. In Proceedings of the IATTC-5th Meeting of the Ad Hoc Working Group on FADs, San Diego, CA, USA, 6 May 2021.
6. Evans, K.; Young, J.; Nicol, S.; Kolody, D.; Allain, V.; Bell, J.; Brown, J.; Ganachaud, A.; Hobday, A.; Hunt, B.; et al. Optimising fisheries management in relation to tuna catches in the western central Pacific Ocean: A review of research priorities and opportunities. *Mar. Policy* **2015**, *59*, 94–104. [CrossRef]
7. Maufroy, A.; Kaplan, D.M.; Bez, N.; De Molina, A.D.; Murua, H.; Floch, L.; Chassot, E. Massive increase in the use of drifting Fish Aggregating Devices (dFADs) by tropical tuna purse seine fisheries in the Atlantic and Indian oceans. *ICES J. Mar. Sci.* **2017**, *74*, 215–225. [CrossRef]
8. Wain, G.; Guery, L.; Kaplan, D.M.; Gaertner, D. Quantifying the increase in fishing efficiency due to the use of drifting FADs equipped with echosounders in tropical tuna purse seine fisheries. *ICES J. Mar. Sci.* **2021**, *78*, 235–245. [CrossRef]
9. Dupaix, A.; Capello, M.; Lett, C.; Andrello, M.; Barrier, N.; Viennois, G.; Dagorn, L. Surface habitat modification through industrial tuna fishery practices. *ICES J. Mar. Sci.* **2021**, *78*, 3075–3088. [CrossRef]
10. Marsac, F.; Fonteneau, A.; Ménard, F. Drifting FADs used in tuna fisheries: An ecological trap? In Proceedings of the Pêche thonière et dispositifs de concentration de poissons, Caribbean-Martinique, France, 15–19 October 1999.
11. Itano, D.; Fukofuka, S.; Brogan, D. The development, design, and current status of anchored and drifting FADs in the WCP. In Proceedings of the 17th Meeting of the Standing Committee on Tuna and Billfish, Majuro, Marshall Islands, 9–18 August 2004; p. 26.
12. Hallier, J.P.; Gaertner, D. Drifting fish aggregation devices could act as an ecological trap for tropical tuna species. *Mar. Ecol. Prog. Ser.* **2008**, *353*, 255–264. [CrossRef]
13. Dagorn, L.; Holland, K.N.; Restrepo, V.; Moreno, G. Is it good or bad to fish with FADs? what are the real impacts of the use of drifting FADs on pelagic marine ecosystems? *Fish Fish.* **2012**, *14*, 391–415. [CrossRef]
14. Moreno, G.; Salvador, J.; Zudaire, I.; Murua, J.; Pelegrí, J.L.; Uranga, J.; Murua, H.; Grande, M.; Santiago, J.; Restrepo, V. The jelly-FAD: A paradigm shift in the design of biodegradable fish aggregating devices. *Mar. Policy* **2022**, *174*, 105352. [CrossRef]
15. Murua, H.; Zudaire, I.; Tolotti, M.; Murua, J.; Capello, M.; Basurko, O.C.; Krug, I.; Grande, M.; Arregui, I.; Uranga, J.; et al. Lessons learnt from the first large-scale biodegradable FAD research experiment to mitigate drifting FADs impacts on the ecosystem. *Mar. Policy* **2023**, *148*, 105394. [CrossRef]
16. Moreno, G.; Salvador, J.; Murua, J.; Phillip, N.B., Jr.; Murua, H.; Escalle, L.; Zudaire, I.; Pilling, G.; Restrepo, V. *A Multidisciplinary Approach to Build New Designs of Biodegradable Fish Aggregating Devices (FADs)*; WCPFC Sci. Comm. WCPFC-SC16-2020/EB-IP-XX; WCPFC: Washington, DC, USA, 2020.
17. Wang, Y.; Zhou, C.; Xu, L.; Wan, R.; Shi, J.; Wang, X.; Tang, H.; Wang, L.; Yu, W.; Wang, K. Degradability evaluation for natural material fibre used on fish aggregation devices (FADs) in tuna purse seine fishery. *Aquac. Fish.* **2021**, *6*, 376–381. [CrossRef]
18. Zhang, T.; Liu, Z.; Zhang, J.; Su, X.; Chen, J.; Wan, R. Numerical Study of the Hydrodynamic Response of Biodegradable Drifting Fish Aggregating Devices in Regular Waves. *Fishes* **2024**, *9*, 112. [CrossRef]
19. Zhou, Y.; Zhang, X.; Chen, J.; Liu, R.; Sun, J.; Si, Y. Catenary Mooring Length Control for Motion Mitigation of Semi-Submersible Floating Wind Turbines. *J. Mar. Sci. Eng.* **2024**, *12*, 628. [CrossRef]
20. Cutajar, C.; Sant, T.; Farrugia, R.N.; Buhagiar, D. Analysis of the Wave Attenuating and Dynamic Behaviour of a Floating Breakwater Integrating a Hydro-Pneumatic Energy Storage System. *J. Mar. Sci. Eng.* **2023**, *11*, 2189. [CrossRef]
21. Wang, S.; Liu, S.; Xiang, C.; Li, M.; Yang, Z.; Huang, B. Prediction of Wave Forces on the Box-Girder Superstructure of the Offshore Bridge with the Influence of Floating Breakwater. *J. Mar. Sci. Eng.* **2023**, *11*, 1326. [CrossRef]
22. Yuan, Y.; Xue, H.; Tang, W. Internal laminar flow effect on the nonlinear dynamic response of marine risers under uniform ocean current. *Ships Offshore Struct.* **2022**, *17*, 1382–1391. [CrossRef]
23. Zhao, Y.P.; Bai, X.D.; Dong, G.H.; Bi, C.W. Deformation and stress distribution of floating collar of net cage in steady current. *Ships Offshore Struct.* **2019**, *14*, 371–383. [CrossRef]

24. Zhang, Y.; Feng, L.; Zhang, L.; Wang, L.; Dong, S.; Gao, Q.; Hu, X.; Li, J. A study of tension and deformation of flexible net under the action of current. *Ships Offshore Struct.* **2022**, *19*, 27–44. [CrossRef]
25. González, M.; Karumathil, S.K. A new method to estimate the resistance coefficients of net panels modeled as porous media. *Ocean Eng.* **2023**, *274*, 113952. [CrossRef]
26. Wan, R.; Zhang, T.; Zhou, C.; Zhao, F.; Wang, W. Experimental and numerical investigations of hydrodynamic response of biodegradable drifting Fish Aggregating Devices (FADs) in waves. *Ocean Eng.* **2022**, *244*, 110436. [CrossRef]
27. Xiao, Y.; Zhou, C.; Zhang, T.; Wang, Y.; Xie, C.; Zhang, Y. Aggregation characteristics of fish aggregating devices (FADs) in tuna purse seine fisheries in the Western and Central Pacific Ocean. *J. Fish. China* **2023**, *47*, 039719.
28. Lee, K.; Lee, S.I.; Kim, D.N. Fluid Dynamics Analysis of Fish Aggregation Device using Particle Image Velocimetry. In Proceedings of the IOTC-2018-WPTT20-24, the 20th Session of the IOTC Working Party on Tropical Tunas, Mahé, Seychelles, 29 October–3 November 2018.
29. Shih, T.H.; Liou, W.W.; Shabbi, A.; Yang, Z.; Zhu, J. A new $k-\epsilon$ eddy-viscosity model for high Reynolds number turbulent flows—Model development and validation. *Comput. Fluids* **1995**, *24*, 227–238. [CrossRef]
30. Fuhrman, D.R.; Li, Y. Instability of the realizable $k-\epsilon$ turbulence model beneath surface waves. *Phys. Fluids* **2020**, *32*, 115108. [CrossRef]
31. Hirt, C.W.; Nichols, B.D. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.* **1981**, *39*, 201–225. [CrossRef]
32. Lin, P.; Liu, P.L.F. A Numerical Study of Breaking Waves in the Surf Zone. *J. Fluid Mech.* **1998**, *359*, 239–264. [CrossRef]
33. Rodi, W. *Turbulence Models and Their Application in Hydraulics—A State of the Art Review*; IAHR Monograph: Delft, The Netherlands, 1980.
34. *STAR-CCM+ User's Guide*; Siemens PLM Software: Plano, TX, USA, 2019.
35. Choi, J.; Yoon, S.B. Numerical simulations using momentum source wave-maker applied to RANS equation model. *Coast. Eng.* **2009**, *56*, 1043–1060. [CrossRef]
36. Gu, H.; Stansby, P.; Stallard, T.; Carpintero Moreno, E. Drag, added mass and radiation damping of oscillating vertical cylindrical bodies in heave and surge in still water. *J. Fluids Struct.* **2018**, *82*, 343–356. [CrossRef]

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

MDPI AG
Grosspeteranlage 5
4052 Basel
Switzerland
Tel.: +41 61 683 77 34

Journal of Marine Science and Engineering Editorial Office

E-mail: jmse@mdpi.com
www.mdpi.com/journal/jmse



Disclaimer/Publisher's Note: The title and front matter of this reprint are at the discretion of the Guest Editors. The publisher is not responsible for their content or any associated concerns. The statements, opinions and data contained in all individual articles are solely those of the individual Editors and contributors and not of MDPI. MDPI disclaims responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.



Academic Open
Access Publishing

mdpi.com

ISBN 978-3-7258-7539-9