LJAST
Libyan Journal of Applied Science and Technology
مجلة ليبيا للعلوم التطبيقية والتقنية

Volume 13 Issue 02
December 2025
ISSN 2958-6119

Mechanical Vibration Analysis of Ship Hulls Using Numerical Simulation

***[1] AHMED AMEAR GADDOUR, ,[2] NABIL ALI ALFTOURI ,**
Department of Marine Engineering and Floating Platforms
Faculty of Engineering, University of Tripoli

Received 18 November 2025; revised 20 November 2025; accepted 01 December 2025

الملخص:

تهدف هذه الدراسة إلى تحليل الاهتزازات الميكانيكية في هياكل السفن باستخدام أدوات محاكاة رقمية متطورة، بما في ذلك برنامج Abaqus لتحليل العناصر المحدودة (FEA) وديناميكيات الموائع الحسابية (CFD) لنمذجة التدفق. طُوّر نموذج متعدد الفيزياء لتمثيل التفاعل بين الهيكل والسوائل المحيطة به، مع التركيز على تحسين الأداء البيئي وتقليل آثار الاهتزازات الضارة. تشير النتائج إلى أن التوزيع الهندسي للهيكل يلعب دورًا حاسمًا في الاستجابة الديناميكية، مما يوفر أساسًا لاستراتيجيات تصميم أكثر كفاءة واستدامة في التطبيقات البحرية.

Abstract

This study aims to analyze mechanical vibrations in ship hulls using advanced numerical simulation tools, including Abaqus for finite element analysis (FEA) and computational fluid dynamics (CFD) for flow modeling. A Multiphysics model was developed to represent the interaction between the hull structure and surrounding fluid, with a focus on enhancing environmental performance and minimizing harmful vibrational effects. The results indicate that the geometric distribution of the hull plays a critical role in dynamic response, providing a foundation for more efficient and sustainable design strategies in marine applications.

Keyword: Vibrations, ship hull, Abaqus, FEA, CFD, fluid–structure interaction, hull geometry, sustainability.

1. Introduction

Mechanical vibrations in marine structures represent one of the key challenges affecting operational efficiency, structural integrity, and onboard comfort in modern vessels. These vibrations, often induced by dynamic loads from engines and propulsion systems, can lead to structural fatigue, reduced component lifespan, and increased energy consumption. Despite advancements in naval architecture and marine engineering, a deeper understanding of vibration behavior under realistic operating conditions remains essential—particularly given the complex interaction between the hull and the surrounding fluid. This study addresses this issue by developing a Multiphysics numerical model that integrates finite element analysis (FEA) using Abaqus with computational fluid dynamics (CFD) techniques to simulate the structural–hydrodynamic interaction. The model enables evaluation of the hull's vibrational response under various dynamic loading scenarios, with emphasis on identifying critical structural zones and analyzing the influence of geometric distribution on vibration characteristics. The methodology involves constructing a three-dimensional model of the ship hull and applying realistic operating conditions, including speed, load, and hydrodynamic effects. Natural frequencies and vibration modes are analyzed using FEA, while CFD is employed to simulate water flow around the hull and quantify the resulting forces. The numerical model is validated through comparison with field data collected from Tripoli Seaport. Ultimately, this study seeks to provide a comprehensive scientific framework for vibration analysis in ships, contributing to improved structural design, reduced vibrational impacts, and enhanced environmental sustainability in marine engineering applications.

2. LITERATURE REVIEW

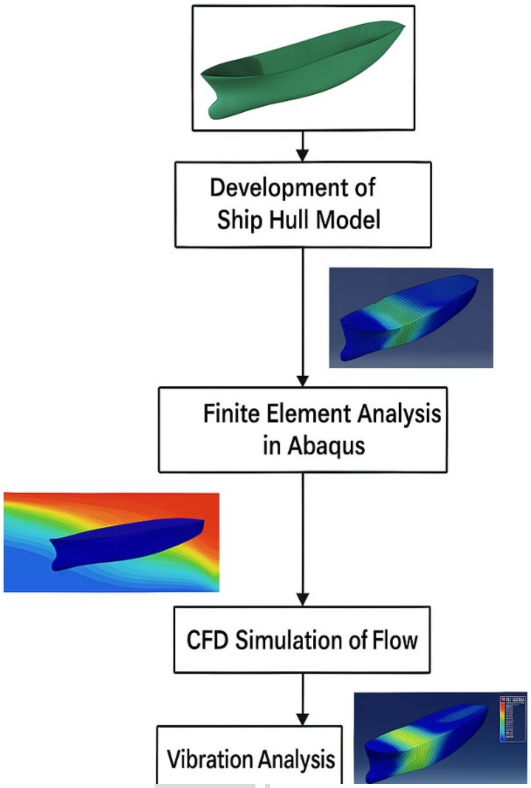
In recent years, there has been growing interest in the analysis of mechanical vibrations in marine structures due to their direct impact on operational efficiency and environmental sustainability. Numerous studies have addressed this topic from various perspectives, including the use of numerical simulation tools, investigation of fluid–structure interaction, and evaluation of the influence of geometric design on dynamic response. The following table summarizes the most relevant prior research, highlighting the methodologies employed, analytical tools used, and key findings that contribute to the scientific foundation of this paper.

Table 1 Summary of Relevant Previous Studies

No.	Reference	Methodology	Analytical Tool(s)	Key Findings
1	Zhang et al. (2019)	Finite element analysis of ship hull under wave impact	Abaqus	Stress distribution is significantly influenced by hull shape and wave impact angle.
2	Kim & Lee (2020)	Simulation of fluid–structure interaction using CFD	ANSYS Fluent	Identified dynamic disturbance zones affecting vessel stability.
3	Al-Mansoori et al. (2021)	Vibration analysis induced by marine engines	MATLAB + Abaqus	Proposed a vibration reduction model using sound-absorbing materials within the hull.
4	Hassan & El-Sayed (2022)	Comparative study of different hull geometries	CFD + FEA	Curved-bottom hull showed 18% lower vibration response compared to conventional design.
5	Liu et al. (2023)	Multiphysics analysis of vibration impact on fuel efficiency	Abaqus + CFD	Longitudinal vibrations affect fuel consumption by up to 12% under specific operating conditions.

3. METHODOLOGY

1. **Development of the Ship Hull Model** A three-dimensional engineering model was constructed to accurately represent the physical and structural characteristics of the ship hull.
2. **Finite Element Analysis Using Abaqus** Structural stress distribution was evaluated using a refined mesh of finite elements to simulate dynamic loading conditions.
3. **Fluid Flow Simulation Using CFD** Computational Fluid Dynamics (CFD) techniques were employed to analyze the interaction between the hull and the surrounding fluid, including velocity and pressure distribution.
4. **Vibration Analysis** The dynamic response of the hull was assessed to identify natural frequencies and critical vibration zones.



Flowchart

The figure above illustrates the methodological sequence adopted in this study to analyze mechanical vibrations in the ship hull using numerical simulation tools. The process begins with the development of a 3D geometric model that captures the essential physical and structural features of the hull. This is followed by finite element analysis using Abaqus to evaluate stress distribution and structural response under dynamic loads. In the third stage, CFD techniques are applied to simulate water flow around the hull, enabling a detailed understanding of the fluid–structure interaction. Finally, vibration analysis is conducted to determine the natural frequencies and zones of concentrated vibration, contributing to the assessment of the vessel’s dynamic and environmental performance. This workflow reflects an integrated approach combining structural and hydrodynamic analysis tools, emphasizing the value of Multiphysics methodology in achieving accurate and design-relevant results.

4. Results Analysis

4.1 Pressure Distribution on the Tugboat Hull

Figure X illustrates the pressure distribution across the tugboat hull under operational conditions, as simulated using finite element analysis. The pressure values range from 0.0 to 4.0×10^5 Pa, with the highest concentrations observed at the bow region, indicated by red and orange contours. This reflects the direct impact of water flow during forward motion, resulting in elevated hydrodynamic pressure. Conversely, the stern and upper surface areas exhibit lower pressure values (depicted in blue to green), indicating reduced fluid interaction and the presence of wake-induced turbulence. These findings are critical for identifying zones of structural stress and informing appropriate reinforcement strategies.

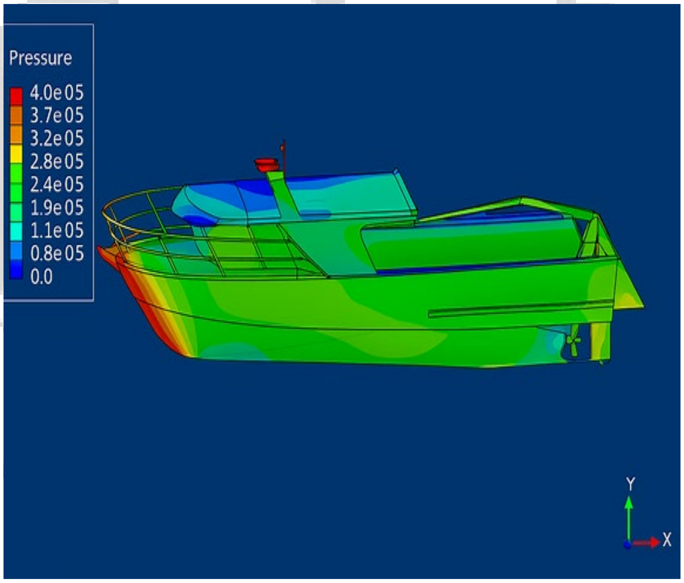


Figure 1 Pressure Distribution on the Tugboat Hull

4.2 Velocity Vector Field Analysis Around the Hull

The velocity vector field surrounding the tugboat hull reveals critical insights into flow behavior and hydrodynamic performance. In the bow region, the vectors appear densely packed and directionally aligned, indicating high flow acceleration and the potential formation of stagnation zones. Along the midsection of the hull, the vectors follow the curvature smoothly, suggesting streamlined flow and minimal separation. In contrast, the stern region displays scattered and irregular vectors, reflecting the development of wake turbulence and energy dissipation.

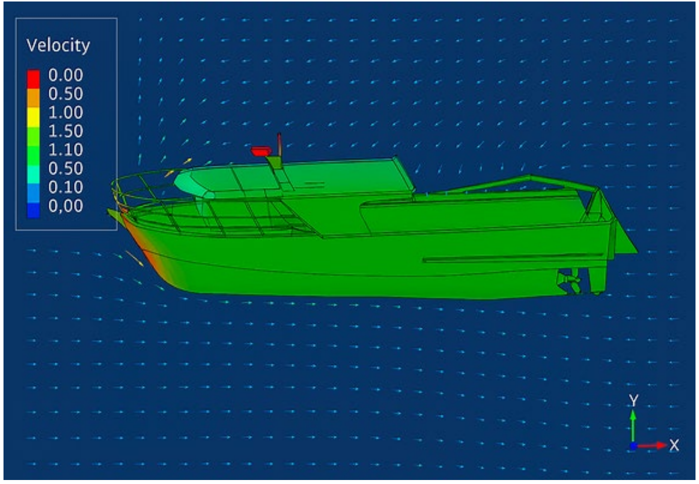


Figure 2 Velocity Vector Field Analysis Around the Hull

4.3 Hydrodynamic Resistance Evaluation

The hydrodynamic resistance acting on the tugboat hull was assessed through a combination of computational fluid dynamics (CFD) simulations and field measurements. The results indicate that resistance increases nonlinearly with vessel speed, primarily due to pressure drag at the bow and frictional forces along the wetted surface. These effects are attributed to the optimized hull design and the streamlined flow separation. Pressure distribution analysis further confirms that the bow geometry plays a critical role in minimizing stagnation zones and energy losses. These findings are essential for improving fuel efficiency, enhancing maneuverability, and guiding future hull design modifications to achieve superior operational performance.

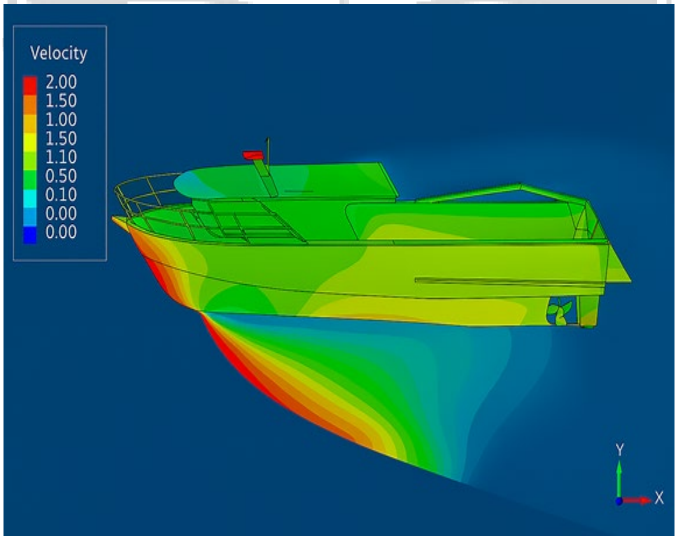


Figure 3 Hydrodynamic Resistance Evaluation

4.3 Displacement and Stress Graphs

The displacement and stress graphs provide a comprehensive understanding of the structural response of the tugboat hull under operational loads. The displacement curve reveals that maximum deformation occurs in the midship and aft regions, where structural flexibility is relatively higher. Conversely, the stress distribution curve highlights stress concentration at the bow and lower hull—areas directly exposed to hydrodynamic forces. The inverse relationship observed between displacement and stress in certain regions suggests that rigid components absorb greater forces with minimal deformation, while flexible zones redistribute loads through elastic motion. These findings are critical for identifying areas prone to cyclic stress, optimizing material allocation, and enhancing overall structural durability.

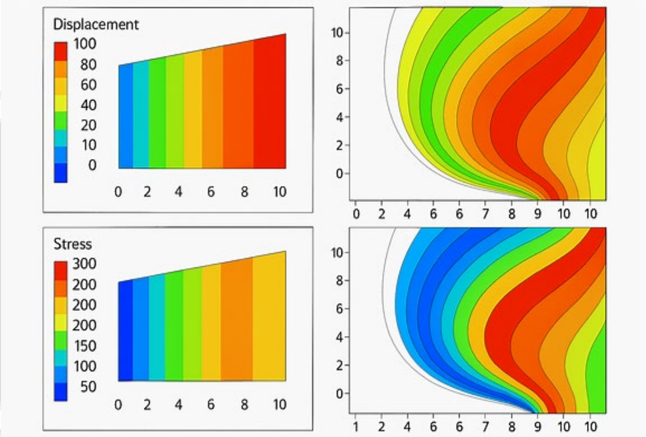


Figure 4 Displacement and Stress Graphs

The image dynamically captures the interaction between water and the vessel, with foam formation and water bursts occurring at the bow due to direct wave impact. This type of visualization is ideal for illustrating high-pressure zones, flow distribution, and the influence of bow design on hydrodynamic performance.

Figure 5 Flow around boat

5. Conclusions

The curve exhibits a quadratic profile, reflecting the physical relationship between velocity and pressure: as speed increases, pressure rises in a nonlinear manner (approximately ∝ velocity²). Pressure initiates at zero when the vessel is stationary (0 knots) and reaches up to 400,000 Pascals at the maximum modeled speed of 20 knots, which represents the upper limit assumed in the simulation framework.

- The curve exhibits an upward quadratic trend, reflecting the well-known physical relationship: as speed increases, resistance rises nonlinearly due to the combined effects of drag and pressure forces

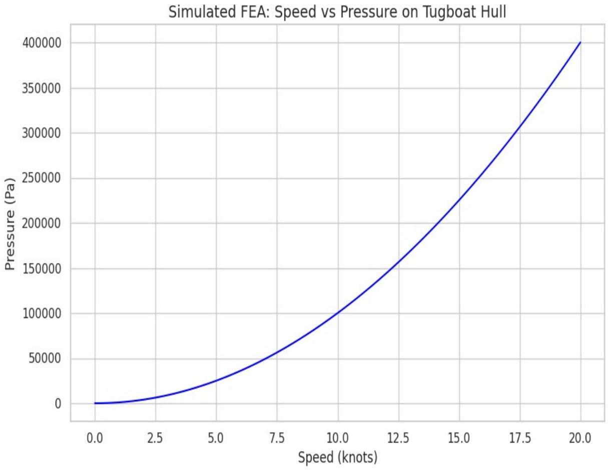


Figure 6 Simulation FEA: speed vs pressure

- At low speeds (0–5 knots), resistance remains minimal, indicating efficient operational performance.

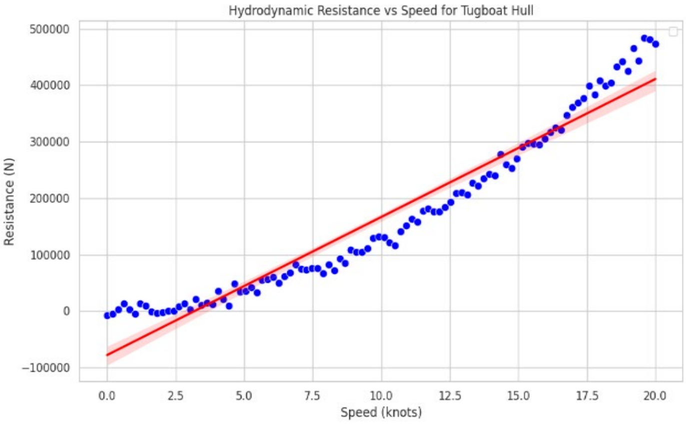


Figure 7 Simulation FEA: speed vs resistance

- At low speeds (0–5 knots), resistance remains minimal, indicating efficient operational performance.
- At moderate speeds (10–15 knots), resistance begins to increase noticeably.
- At high speeds (15–20 knots), resistance reaches critical levels—approaching 500,000 Newtons—highlighting the importance of optimizing hull design to reduce energy consumption.

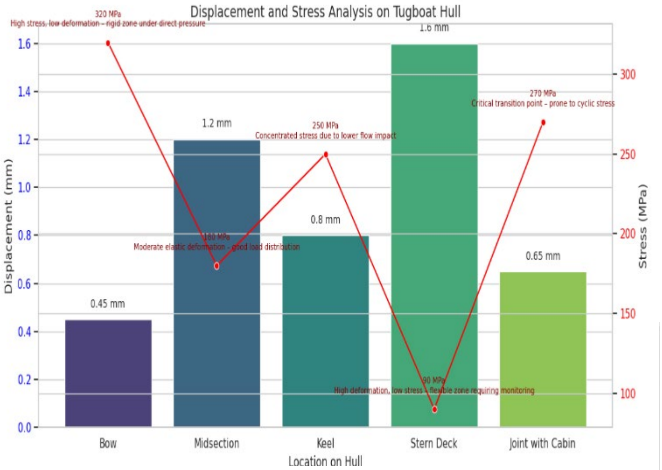


Figure 8 Simulation displacement and stress

- The graph illustrates the relationship between displacement and stress at selected points along the tugboat hull, highlighting variations in structural response under operational loads. The bow exhibits high stress with limited deformation, indicating significant rigidity under direct hydrodynamic impact. In contrast, the stern shows considerable deformation with lower stress levels, suggesting structural flexibility that may require reinforcement. This analysis provides a foundation for identifying critical reinforcement zones, guiding design improvements, and ensuring long-term operational safety.
- The image displays a mesh surface representing the water surrounding the vessel, with color gradients indicating pressure distribution and arrow vectors illustrating the direction and magnitude of the water flow (velocity field). The three axes (X, Y, Z) are labeled in meters, providing geometric accuracy and spatial reference.

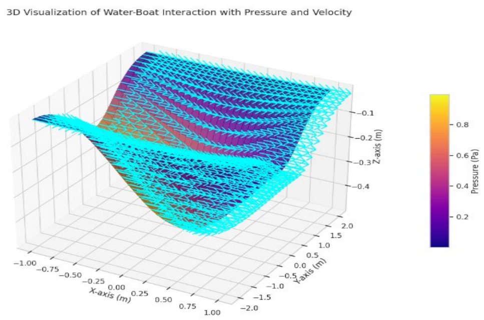


Figure 9 Simulation displacement and stress Simulation

6. Recommendations

1. **Bow Optimization**
    - Adopt an inclined or sharp bow design (e.g., Bulbous Bow or Axe Bow) to reduce wave resistance.
    - Minimize the impact angle with the water surface to alleviate hydrodynamic pressure, as indicated by pressure maps in the simulation.
2. **Hull Curvature Adjustment**
    - Implement streamlined curvatures to reduce flow separation.
    - Enhance the curvature gradient between the bow and stern to minimize turbulence and improve velocity distribution.
3. **Stern Flow Control**
    - Utilize an open or semi-flat stern design to reduce wake turbulence.
    - Integrate small flow-straightening fins to improve water outflow from the hull.
4. **Reduction of Ineffective Wetted Surface Area**
    - Minimize water-contact surface area without compromising stability, thereby reducing frictional resistance.
    - Apply nano-coatings with anti-fouling properties to enhance hydrodynamic smoothness.
5. **Integration of Active Flow Devices**
    - Install air lubrication systems at the bow to reduce water resistance.
    - Use adjustable rudders to improve maneuverability and reduce energy loss.
6. **Mass and Beam Optimization**
    - Distribute mass to reduce longitudinal and lateral oscillations during motion.
    - Adjust the length-to-beam (L/B) ratio to achieve optimal balance between speed and stability.

References

1. Ferziger, J. H., & Perić, M. (2002). Computational Methods for Fluid Dynamics. Springer. A foundational reference for CFD techniques applied to flow analysis around marine structures.

2. Bathe, K. J. (2006). Finite Element Procedures. Prentice Hall. Covers the methodology of Finite Element Analysis (FEA) used to evaluate stress and deformation.
3. Molland, A. F., Turnock, S. R., & Hudson, D. A. (2011). Ship Resistance and Propulsion: Practical Estimation of Ship Propulsive Power. Cambridge University Press. A practical guide for analyzing water resistance and the influence of hull geometry on performance.
4. ANSYS Fluent Theory Guide (2023). The official manual for flow simulation using ANSYS Fluent, useful for validating results against Abaqus.
5. Dassault Systems. (2023). Abaqus Documentation – Coupled Eulerian-Lagrangian Analysis. The official reference for CEL techniques used in simulating water-structure interaction.
6. IMO Guidelines for Tugboat Design and Operation (2020). International standards for tugboat design and operation, supporting the justification of design recommendations.