The International Maritime Transport and Logistics (MARLOG 14) - ISSN 2974-3141
http://dx.doi.org/10.21622/MARLOG.2025.14.1.72
Artificial Intelligence Implementations Towards Shaping the Future of Digital World

ENHANCING MARITIME SAFETY: OPTIMIZING FAIRWAY DESIGN THROUGH MULTI-ARC CONFIGURATIONS

Miral Michel (1), Gerges Refaat Fawzy (1), Youssef Wael Hamed (1), Zeyad Ayman Soliman (1), Mohamed Elnegouly (1), Ahmed Mehanna (1) and Ahmed S. Shehata (1)

(1) Marine and Offshore Engineering Department, Arab Academy for Science, Technology and Maritime Transport, Alexandria, Egypt

ABSTRACT

The growing parameters of ships entail the necessity for port infrastructure which depends mainly on the size of ships and various weather conditions. One of the major problems in marine traffic engineering is to determine the safe parameters of fairways, i.e., safe depth and width at the bottom in bends of the fairway. While examining the velocity component of bends, the helical flow formed by secondary currents was observed. A secondary current occurs due to imbalance between centrifugal forces and pressure gradient at surface. In other words, close to the inner bank and at channel bed, pressure gradient exceeds centrifugal forces and conveys water in transverse direction toward the inner bank. At free surface, centrifugal forces drive flow (secondary current) to the outer bank. The aim of this research is to replace fairway bends that have a single arc with multiple separate arcs to reduce the curvature effect of fairways on ships during manoeuvring. This is achieved by performing CFD methods for fairway bends, with a single arc shape and with multiple separate arcs, under different flow conditions. Multiple arcs will reduce the effect of helical flow with the objective of minimizing grounding risk, loss of steering, or ships collision.

Keywords: Fairways, Navigation, Safety, CFD

1. INTRODUCTION

Autonomous vessels and the future of maritime transport raise questions about the need for smarter fairways. Maritime transport remains a vital mode of transportation, with approximately 90% of global shipping occurring by sea [1]. Over time, advancements in shipping technology have made maritime transport more efficient and safer. Modern ships have eliminated distance and time constraints, allowing for convenient global travel [2]. Fairways play a crucial role in maritime transportation, providing safe navigation routes through bodies of water [3]. Marked by buoys, beacons, and lights, fairways minimize the risk of accidents like collisions and groundings. They also help manage maritime traffic, particularly in congested areas, ensuring the smooth flow of vessels and optimal use of infrastructure [4]. Additionally, well-defined fairways contribute to the protection of the marine environment by minimizing accidental damage to sensitive habitats and ecosystems [5]. Effective fairways facilitate the efficient operation of ports and harbors, allowing for organized cargo handling and timely trade. Compliance with international standards, like those set by the International Maritime Organization (IMO), promotes global maritime safety, connectivity, and cooperation among stakeholders [6], [7]. Therefore, as maritime transport continues to evolve, smarter fairways will be essential to ensure safe and efficient maneuverability [8].

The current state of fairways poses significant challenges for the navigation and transportation of vessels. These challenges include inadequate dredging and maintenance, limited channel depth and width, poor buoyage systems, restricted access to ports and harbors, navigational hazards, and insufficient infrastructure for vessel maneuvering and berthing. These issues lead to increased risks of accidents, shipping delays, higher maintenance costs, and reduced capacity for trade and economic growth [5]. For example, sedimentation can make fairways shallower, hindering the passage of larger vessels and limiting port capacity. Improperly maintained or positioned buoys can mislead ship operators and result in accidents. Inadequate fairway infrastructure can restrict vessel size and impede economic growth. Navigational hazards, if not marked or addressed, can cause accidents and environmental damage. Inadequate infrastructure for vessel maneuvering and berthing can lead to congestion and delays. Addressing these challenges is crucial to ensure safe maritime traffic, optimize port efficiency, and support international [9].

In a previous paper by Ohtsu, Yoshimura *et al.* [10], the need for a more sophisticated design standard for fairways was discussed. The research aimed to establish a practical design standard for the depth, width, and alignment of fairways in port water facilities to ensure safe and efficient navigation for ships. The proposed standard can also be used to evaluate existing fairways and provide guidance for shipbuilding. The research outcomes are expected to enhance navigation safety within port water facilities.

In a more recent article by Lebedev and Lipatov [11], the shortcomings of existing methods used to determine safe fairway widths were investigated. Their proposed solution involved the development of a more rigorous mathematical model that would incorporate a wider range of factors, including variations in rudder control and engine parameters. The research further emphasizes the importance of robust simulations for training navigators and calls for the revision of current regulations to reflect advancements in navigational practices and technologies.

Furthermore, Stancu's investigation [12] into sediment transport within the Danube River aimed to identify strategies for optimizing fairway reliability. Stancu employed a sophisticated hydraulic model to simulate the river's behavior over a four-month period. This simulation allowed for the exploration of various sediment management scenarios, paving the way for the development of more effective measures to ensure safe and reliable navigation routes.

Another study by Gucma and Zalewski [13] focused on determining safe maneuvering areas for ships on waterways. They used simulations to provide detailed design and safe operation conditions, specifically on the Świnoujście-Szczecin fairway reconstruction. Different methods, including empirical and deterministic approaches, were used to determine safe maneuvering widths.

Yoshimura [14] investigated the technique of maneuvering ship motion in shallow water using a mathematical model. The study demonstrated the practical application of the model in shallow water conditions, improving the estimation formula for ship sinking and trim.

Lastly, Gucma, Przywarty *et al.* [15] presented a kinematic method for determining safe fairway bend width. The method estimates safe bend width based on turn angle and provides more accuracy compared to other empirical methods.

While these advancements offer strategies for managing helical flow, there is still a gap in developing a permanent solution. Existing research primarily focuses on mitigating the effects of helical flow, but a more universal solution is needed to ensure navigation safety and efficiency. Helical flow can lead to increased risks of grounding and collisions, delays in navigation, and limitations on ship size and type.

To address this gap, this research initiative centers around the implementation of a novel design concept for waterway bends, specifically targeting a reduction in the curvature's influence on ships during maneuvering operations. The case study for this project focuses on the Suez Canal, a vital maritime artery in Egypt. The core idea lies in replacing the traditional single-arc bends currently used in fairway design with a configuration comprised of multiple, distinct arcs.

To achieve this objective, the research team employed Computational Fluid Dynamics (CFD) methods. These simulations were conducted on four variations of fairway bends: one featuring a conventional single arc shape, While the other design depends on dividing the arch into several arches, reaching two, three, or four. Multiple separate arcs will reduce the effect of helical flow. This phenomenon arises due to the presence of secondary currents within the waterway bend. By

http://dx.doi.org/10.21622/MARLOG.2025.14.1.72

implementing a series of separate arcs, the research team anticipates a significant reduction in the intensity of this helical flow. This translates to a tangible decrease in the risk of ships encountering grounding incidents, a loss of steering control, or even collisions during maneuvers within the bend [16].

2. METHODOLOGY

This section details the computational approach used in this research to evaluate the influence of replacing a single arc fairway with a multi-arc design on water entry behavior, force, velocity, and pressure distribution on a model ship.

2.1. CFD Numerical Model

The commercial CFD software, ANSYS Fluent, was employed for numerical simulations. The Reynolds-Averaged Navier-Stokes (RANS) equations were utilized to model the turbulent flow, along with a suitable turbulence mode.

$$\rho\left[\frac{\partial u}{\partial t}+u.(\nabla u)\right]=-\nabla p+\mu\nabla^2 u \qquad (1)$$

$$\nabla.u=0 \qquad (2)$$

Eq. (1) is the momentum equation and Eq. (2) is the mass conservation equation, in which $u(x,t)$ is the flow velocity field, x is the vector of spatial coordinates and t is the time, ρ is constant mass density, $p(x,t)$ is the flow pressure, μ is the dynamic viscosity, x=$xkek$ are the position coordinates, ekek is a base vector, u is the flow velocity, ∇=ek ∂/∂xk is a gradient operator, and $\nabla 2(:)=\nabla\cdot\nabla(:)$. The motion of an incompressible Newtonian fluid satisfies the continuity equation in Eq. (3) and conservation of momentum equation in Eq. (4).

$$\frac{\partial(u_i)}{\partial x_i}=0 \qquad (3)$$

$$\frac{\partial u_i}{\partial t}+\frac{\partial u_i u_j}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\mu\frac{\partial u_i}{\partial x_j}\right)-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+S_j \qquad (4)$$

where ui and uj are the time mean of the velocity component (i, j = 1, 2, 3), p is the time mean of the pressure, ρ is the fluid density, μ is the dynamic viscosity coefficient, and Sj is the generalized source term of the momentum equation.

2.2. Geometry and mesh generation

A two-dimensional (2D) model of the single-arc and multi-arc fairway configurations was created. The multi-arc design consisted of variations with two, three, and four arcs (one arc was designed and then divided into two identical arcs by making a rectangular hole in the side of the canal. This way, three arcs and four arcs were designed for study). A structure was generated for each geometry, ensuring appropriate mesh refinement near the ship and around areas of complex flow near the arc transitions as shown in figure (1).

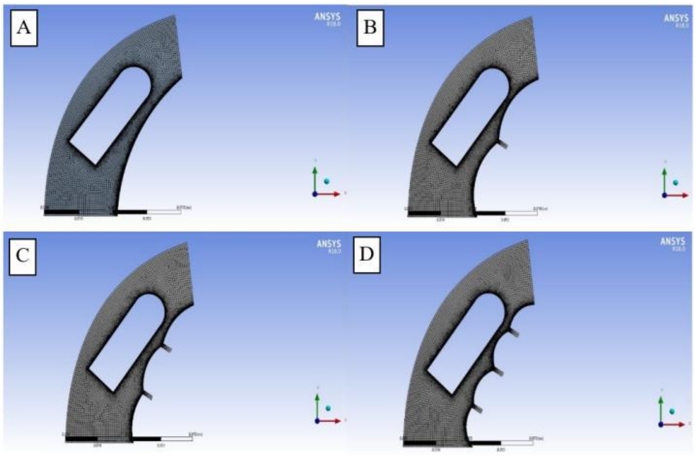

Figure 1: Mesh generation for: (A) single arc, (B) two arcs, (C) three arcs, and (D) four arcs.

A meticulous mesh refinement study was conducted to guarantee the accuracy of our simulations. This involved iteratively generating meshes with increasing cell densities and evaluating the solutions for stability. While the results appeared to converge at approximately 30,000 cells, we opted to continue refining the mesh to achieve a targeted average y+ value between 1.7 and 1.8. This specific y+ range ensures a well-resolved boundary layer near the ship's hull, which is crucial for capturing the intricate flow physics in this region. The boundary layer significantly influences phenomena like drag and lift forces acting on the ship, and a well-refined mesh in this area is essential for obtaining reliable predictions.

2.3. Boundary conditions

For the water inlet, a velocity inlet boundary condition was implemented. This condition specifies a uniform velocity profile of 3 m/sec at the canal entrance. The pressure at the inlet is set to the ambient atmospheric pressure, which is typically close to zero-gauge pressure (1 atm).

2.4. Solution methods

For pressure-velocity coupling, the SIMPLE algorithm (SIMPLE) was employed. This pressure-based solver iteratively solves the momentum and pressure equations to achieve mass conservation. For spatial discretization, a least squares cell-based method was utilized for the gradient calculations. This approach provides a good balance between accuracy and computational efficiency.

The governing equations for pressure, momentum, turbulent kinetic energy (k), specific dissipation rate (ω), intermittency (γ), and momentum thickness Reynolds number ($Re\theta$) were all discretized using a second-order upwind scheme. This scheme offers improved accuracy in capturing convective terms in the equations, particularly important for turbulent flows.

2.5. Validation of the CFD model

A comprehensive computational fluid dynamics (CFD) investigation was undertaken to validate the hydrodynamic performance of a ship model under specific operating conditions. The numerical simulations were conducted within a meticulously defined rectangular computational domain that closely mirrored the physical dimensions of the experimental towing tank at the University of Liege as shown in Fig. (2). To ensure accurate representation of experimental conditions, the CFD model was subjected to identical flow velocities and environmental factors as those encountered in the physical tests.

Figure 2: Experimental towing tank of University of Liege.

The ship, maintained in a fixed position within the confined channel to replicate experimental constraints, was subjected to a range of speeds spanning from 0.335 m/sec to 0.575 m/sec. The primary focus of the study was to quantify and compare the hydrodynamic resistance generated by the ship, as determined through both CFD simulations and experimental measurements. Achieving convergence for each simulated speed regime necessitated substantial computational resources and time. Upon convergence, a detailed analysis was performed to correlate the simulated resistance values with the corresponding experimental data. The overall objective of this research was to establish the credibility and predictive capabilities of the CFD model for accurately simulating ship hydrodynamics. To expedite the computational process and simplify the analysis, the experiment was replicated in a two-dimensional (2D) environment using ANSYS CFD as illustrated in Fig. (3). To accurately represent the physical phenomenon, meticulous attention was paid to establishing appropriate boundary conditions that aligned with the experimental setup. Given the intricate geometric features of the model, a high-quality mesh was generated using advanced meshing techniques to capture the complex flow patterns accurately. Particular emphasis was placed on refining the mesh within the boundary layer region to precisely resolve the near-wall flow characteristics, which are crucial for determining forces and flow properties. To ensure consistent time-accurate solutions, a fixed time step of 0.05 seconds was implemented for all simulations.

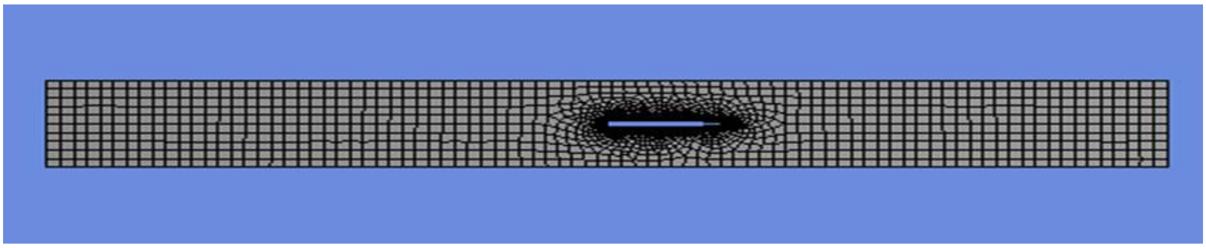

Figure 3: Computational domain of the validation simulation.

A comparison of the simulation and experimental data, presented in Table (1), indicates a strong correlation with discrepancies in drag forces values remaining within a 4% margin across all test conditions. While the CFD model consistently overestimates resistance compared to experimental measurements, this deviation falls within the expected range of experimental error, validating the model's overall accuracy.

Table 1. Comparison of drag force between simulation and experiment

Ship speed (m/sec)	*Simulation drag force (N)*	*Experimental drag force (N)*	*Error (%)*
0.335	1.35	1.318	**2.4279**
0.4485	2.48	2.388	**3.7096**
0.575	**4.128**	**4.248**	**2.8248**

After this validation, the curvature of the channel and the size of the ship were changed to match the research hypothesis.

3. RESULTS

3.1. Water pressure distribution and impact on ship performance

Case 1 (base case): This case serves as the baseline for comparison. The pressure distribution across the hull was relatively even, indicating minimal lateral forces acting on the ship and a balanced trim as shown in Fig. (4). However, it is clear that there are some areas around the ship where the pressure coefficient drops to -4.6, while in the opposite direction it reaches 5.

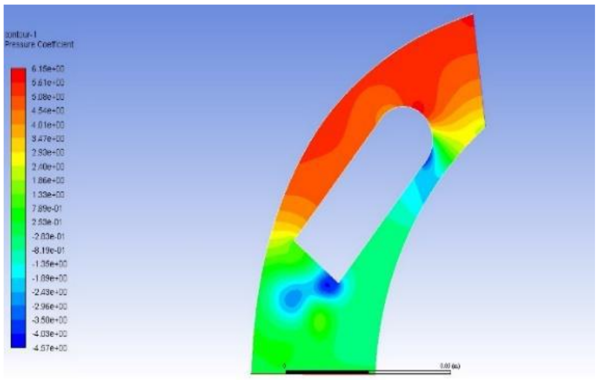

Figure 4: Pressure distribution on the single arc design.

Case 2: Compared to case 1, case 2 exhibited an improvement in pressure distribution. While not entirely symmetrical, the pressure difference between the inner and outer sides of the curved section was less pronounced. This translates to reduced lateral forces acting on the ship, making it easier to maintain course as shown in Fig. (5). In this case, it becomes clear that some low-pressure coefficient areas disappear, reaching -1.5 while on the other side of the ship, it is observed that the pressure increases slightly, reaching 5.5.

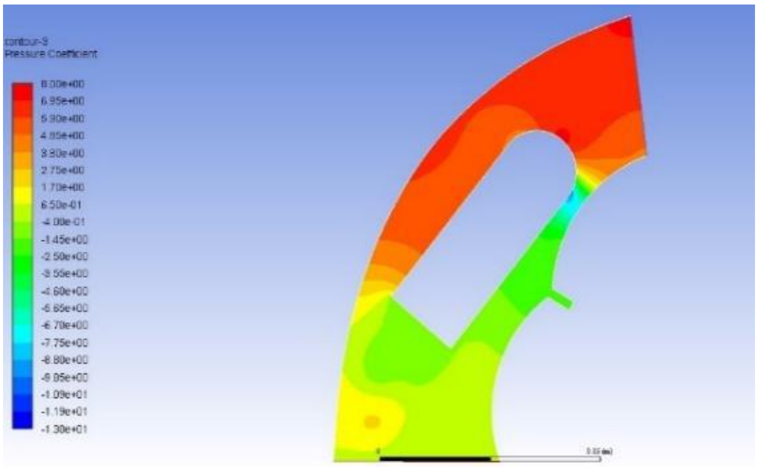

Figure 5: Pressure distribution on the two arcs design.

Case 3: This case demonstrated the most favorable pressure distribution among the curved fairways (cases 2-4). The pressure differential between the sides was further minimized compared to cases 1 and 2 where the pressure coefficient on the low side rises to reach -0.4 and all low-pressure points disappear and the pressure coefficient on the other side decreases to reach 5. This configuration would likely result in the least resistance and the most balanced trim for the ship navigating the curve as shown in Fig. (6).

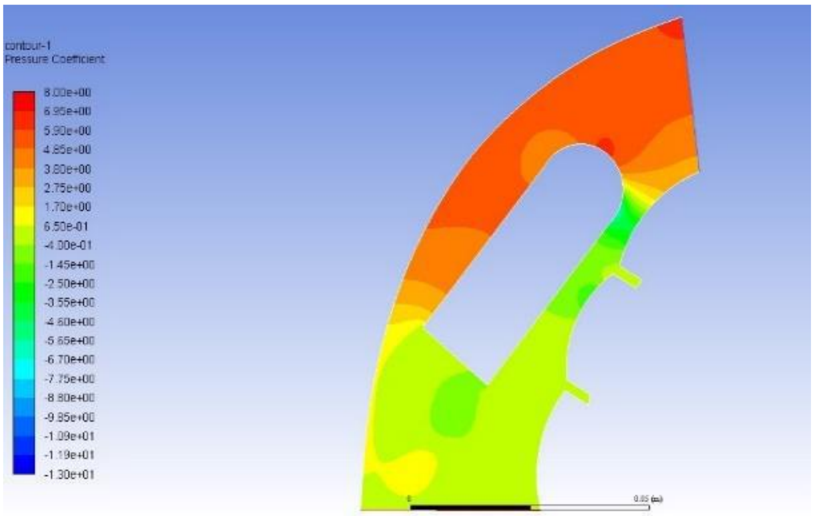

Figure 6: Pressure distribution on the three arcs design.

Case 4: Although an improvement over case 1, case 4 showed a slightly higher pressure-difference than case 3 where the pressure coefficient decreases again until it reaches -1.5 symmetrically on one side of the ship, and the pressure coefficient rises on the other side to reach 7. This indicates potential for a larger lateral force acting on the ship as it navigates the curve. While still manageable, this case might require a slightly larger rudder correction compared to case 3 to maintain course as shown in Fig. (7).

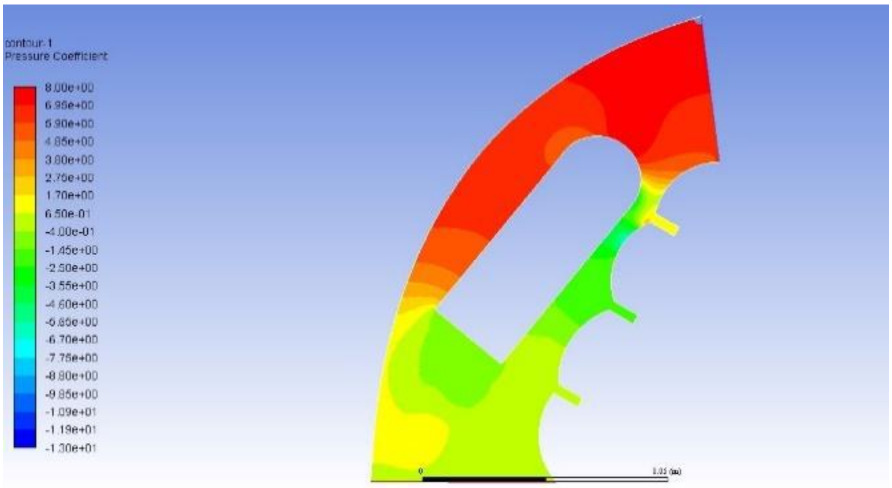

Figure 7: Pressure distribution on the four arcs design.

Overall, the analysis suggests that the pressure distribution in case 3 offered the most optimal scenario for ship performance within the curved fairway. cases 2 and 4 showcased improvements compared to the base case (case 1), but case 3 achieved the most balanced pressure distribution, minimizing lateral forces and resistance.

3.2. Velocity distribution

This analysis of fluid velocity distribution aligns with the previous findings on pressure distribution. Uneven velocity creates areas of high and low pressure, ultimately influencing the overall pressure distribution around the ship. This confirms the prior findings: a smooth and even velocity distribution (case 3) leads to a balanced pressure distribution due to minimal variations in forces acting on the hull, uneven velocity distribution in fairways (case 1) creates pressure differences due to faster flow on the outside (lower pressure) and potentially slower flow on the inside (higher pressure), and the optimized curvature of case 3 minimizes velocity variations, resulting in a pressure distribution to the ideal scenario as shown in Fig. (8).

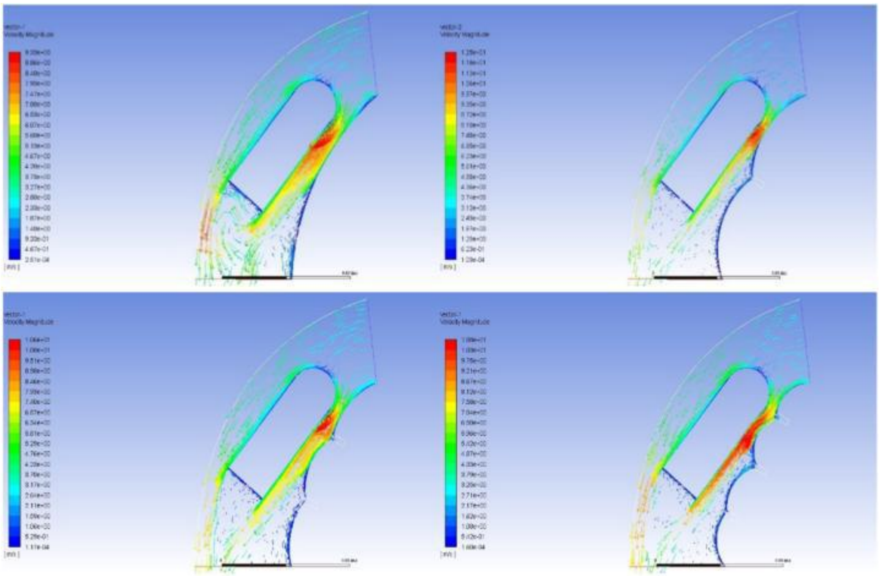

Figure 8: Velocity distribution on: (A) single arc, (B) two arcs, (C) three arcs, and (D) four arcs designs.

3.3. Success of the three-arc design

The findings from the drag shown in Fig. (9) and lift as shown in Fig. (10) analysis strongly support the concept of a three-arch fairway design (3 arcs) for facilitating ship maneuvering. By minimizing drag force, this design allows for smoother and more efficient movement within the curved channel. This translates to improved maneuverability and potentially reduces fuel consumption due to lower resistance.

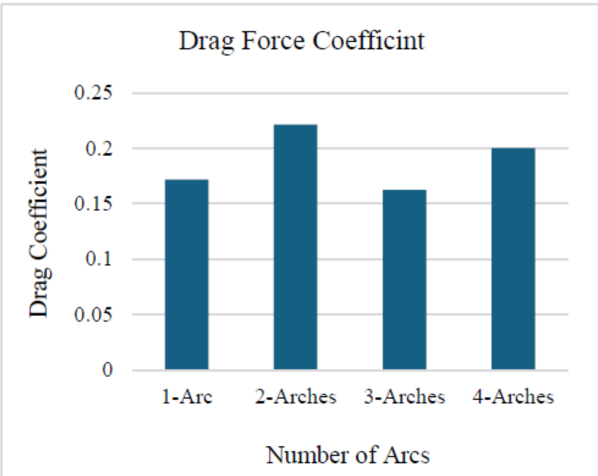

Figure 9: Drag Force Coefficient for the four arcs designs.

http://dx.doi.org/10.21622/MARLOG.2025.14.1.72

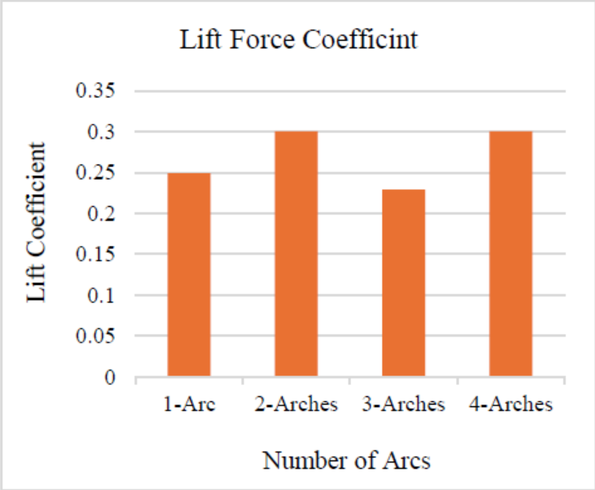

Figure 10: Lift Force Coefficient for the four arcs designs.

Overall, the CFD simulations demonstrate that the three-arc design offers the most favorable scenario for ship maneuverability by minimizing drag force. This finding validates the effectiveness of this approach in facilitating ship navigation through curved navigational channels.

4. CONCLUSIONS

This study has provided a comprehensive investigation into optimizing fairway design through the implementation of multi-arc configurations, aiming to enhance maritime safety and operational efficiency. The results clearly demonstrate that the three-arc configuration offers superior performance, achieving a 12% reduction in drag force compared to the traditional single-arc design and reducing pressure differentials by 44%, thereby ensuring a more balanced and stable distribution across the hull. These advancements translate into significant practical benefits, including enhanced navigational safety, reduced risk of collisions and groundings, and improved efficiency in maritime operations. In critical waterways such as the Suez Canal, where traffic density and safety are paramount, adopting this design could play a pivotal role in minimizing congestion and operational hazards, while also contributing to the long-term sustainability of maritime transport systems. While the CFD simulations provide robust and credible insights, real-world validation under varied environmental and operational conditions is essential to solidify these findings. Furthermore, future research should explore the long-term implications of this design, including sedimentation management, environmental adaptability, and its potential integration into international maritime standards. This study sets a foundation for innovative fairway designs that prioritize safety, efficiency, and sustainability in global maritime operations

5. REFERENCES

1. T. Notteboom, A. Pallis, and J.-P. Rodrigue, Port Economics, Management and Policy. 2022.
2. U. N. C. o. Trade and Development, Review of Maritime Transport 2022. United Nations, 2022.
3. R. Mehdi, J.-U. Schröder-Hinrichs, and M. Baldauf, "A Tale of Two Industries: Seafarer Perceptions of Navigational Safety Risks Near Offshore Wind Farms," Ocean Yearbook, vol. 33, pp. 543-581, 05/01 2019, doi: 10.1163/9789004395633_021.
4. United Nations, "Review of Maritime Transport 2023," 2023.
5. V. Koilo, "Sustainability issues in maritime transport and main challenges of the shipping industry," Environmental Economics, vol. 10, pp. 48-65, 09/03 2019, doi: 10.21511/ee.10(1).2019.04.
6. UNCTAD, "UNCTAD Handbook of Statistics 2023," 2023.

7. IMO, "2023 IMO Strategy on Reduction of GHG Emissions from Ships," 2023.
8. J. W. F. Remo, A. Khanal, and N. Pinter, "Assessment of chevron dikes for the enhancement of physical-aquatic habitat within the Middle Mississippi River, USA," Journal of Hydrology, vol. 501, pp. 146-162, 2013/09/25/ 2013, doi: https://doi.org/10.1016/j.jhydrol.2013.07.007.
9. V. V. Karetnikov, A. A. Butsanets, A. I. Zaytsev, and E. A. Ratner, "Gradient fill fairway plotting method for mapping inland waterways," IOP Conference Series: Earth and Environmental Science, vol. 867, no. 1, p. 012011, 2021/10/01 2021, doi: 10.1088/1755-1315/867/1/012011.
10. K. Ohtsu, Y. Yoshimura, M. Hirano, H. Takahashi, and M. Tsugane, "Design standard for fairway in next generation," vol. 20, p. 6, 09/01 2008, doi: 10.14856/kanrin.20.0_6.
11. O. Lebedev and I. Lipatov, "Determining the width of the ship's course using a mathematical model to ensure the safety of the ship's navigation," Journal of Physics: Conference Series, vol. 2131, p. 032034, 12/01 2021, doi: 10.1088/1742-6596/2131/3/032034.
12. M. Stancu, "Hydrodynamic and sediment transport modelling for channel bed stabilization," IOP Conference Series: Earth and Environmental Science, vol. 1185, p. 012020, 05/01 2023, doi: 10.1088/1755-1315/1185/1/012020.
13. S. Gucma and P. Zalewski, "Optimization of fairway design parameters: Systematic approach to manoeuvring safety," International Journal of Naval Architecture and Ocean Engineering, vol. 12, pp. 129-145, 2020/01/01/ 2020, doi: https://doi.org/10.1016/j.ijnaoe.2019.08.002.
14. Y. Yoshimura, "Mathematical model for the manoeuvring ship motion in shallow water," Journal of the Japan Society of Naval Architects and Ocean Engineers, vol. KASNAJ, pp. 41-51, 01/01 1986.
15. S. Gucma, M. Przywarty, J. Dzwonkowski, and M. Bilewski, "Dimensioning of Fairway Bends—Kinematic Method of Numerical Simulation," Journal of Marine Science and Engineering, vol. 8, no. 2, doi: 10.3390/jmse8020138.
16. A. Haider, "Drag Coefficient and Terminal Velocity of Spherical and Non-Spherical Particles," Powder Technology - POWDER TECHNOL, vol. 58, pp. 63-70, 05/01 1989, doi: 10.1016/0032-5910(89)80008-7.