International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 12 Issue 5 ǁ May 2024 ǁ PP. 345-352

Integration of CFD Methodology for Ship Air Resistance Assessment - A Practical Approach

Nguyen Manh Chien [*1]
[*1]*Department of Shipbuilding, Vietnam Maritime University, Vietnam*
Corresponding Author: Nguyen Manh Chien

Abstract
Air resistance is a critical factor influencing the performance and maneuverability of ships with large main deck structures, such as container ships, liquefied natural gas (LNG) tankers, and large cruise ships. This study aims to provide a comprehensive analysis of air resistance on LNG tankers by employing computational fluid dynamics (CFD) methods. Given the high computational time required for CFD calculations, this study focuses on practical aspects to streamline the process. The CFD results are meticulously validated through experimental tests conducted in a wind tunnel, ensuring accuracy and reliability. By comparing the CFD results with empirical data, this study not only highlights the efficacy of CFD in predicting air resistance but also underscores the importance of integrating experimental validation in aerodynamic studies of large vessels. The findings from this research offer valuable insights into optimizing ship design to minimize air resistance, thereby enhancing overall efficiency and operational performance.

Keywords: *CFD, air resistance, practical approach, wind tunnel, optimization, aerodynamics.*

Date of Submission: 16-05-2024 Date of acceptance: 31-05-2024

I. INTRODUCTION

In recent years, the trend in shipbuilding has leaned towards constructing larger and more robust vessels. As ships increase in size, the impact of wind load becomes a critical factor, particularly when these vessels maneuver in high wind conditions or navigate through confined spaces such as canals and narrow straits. A notable incident underscoring the significance of wind load occurred in March 2021, when the container ship Ever Given ran aground in the Suez Canal. High winds played a pivotal role in this event, severely compromising the ship's maneuverability and leading to a significant disruption in global trade. Despite the evident importance of wind resistance, it remains an area that has not been extensively covered in theoretical studies, and empirical data from wind tunnel testing is relatively sparse.

With the advancement of high-speed computer systems, Computational Fluid Dynamics (CFD) methods have become increasingly viable and are now commonly applied to solve complex hydrodynamic and aerodynamic problems. CFD allows for detailed simulations that can predict the behavior of airflows around ship structures, offering insights that are difficult to obtain through traditional experimental methods alone. In this context, W.D. Janssen et al. [1] conducted a noteworthy study that calculated wind resistance for container ships using CFD and compared the results with tunnel model testing, demonstrating the potential of CFD in maritime applications.

However, there is a gap in the literature when it comes to LNG tankers. These vessels, with their expansive surface areas and unique structural features, are significantly affected by air resistance. Despite this, few studies have focused on testing and comparing wind resistance results for LNG tankers using both CFD and experimental methods. This oversight presents an opportunity to further explore and validate the efficacy of CFD in predicting wind loads for LNG tankers, ultimately contributing to more efficient and safer ship designs. This study addresses this gap by presenting a comprehensive analysis of wind resistance on LNG tankers, utilizing both CFD simulations and wind tunnel experiments to provide robust and validated results.

Furthermore, the approach outlined in this study is designed to bridge the gap between theoretical models and real-world applications, ensuring that the CFD method is not only accurate but also feasible for practical use. The paper discusses the step-by-step process of setting up CFD simulations, including the selection of appropriate boundary conditions, mesh generation, and turbulence modeling. Special attention is given to optimizing computational resources to reduce the time and cost associated with extensive CFD calculations, which is often a significant barrier to its widespread adoption in the maritime industry. By integrating empirical data from wind tunnel experiments, the study provides a comprehensive validation framework that enhances the credibility and reliability of the CFD results. Thes pragmatic approach ensures that the CFD methods proposed are not merely academic exercises but are applicable to actual ship design and performance analysis, thereby offering valuable insights and tools for naval architects and marine engineers.

II. CFD CALCULATION SETUP

2.1. Ship model

First, the 3D model of the LNG tanker was meticulously constructed using Rhinoceros software, a powerful tool widely used in naval architecture for its precision and versatility. The authors created two distinct architectural designs for the main deck of the ship to explore different structural impacts on air resistance: one featuring a spherical cargo tank and the other featuring a box-shaped cargo tank. This dual approach allowed for a comparative analysis of how different tank geometries affect aerodynamic performance.

The basic parameters of the ship, including dimensions such as length, breadth, draft, and tank capacity, are detailed in Table 1. These parameters provided the foundational data necessary to ensure that the 3D models accurately represented the real-world ship. The dimensions were carefully scaled down to a ratio of 1:251, resulting in a physical model with an overall length of 1 meter. This scale was chosen to maintain the fidelity of the model while making it manageable for wind tunnel testing.

The construction of the 3D models involved a high level of detail to capture the significant features of the LNG tanker. Once the digital models were completed, the physical models were fabricated using a combination of wood and plastic materials, with certain components produced via 3D printing to ensure precision. The models were then polished to achieve a smooth surface finish, which is crucial for accurate aerodynamic testing.

This meticulous process ensured that the physical models were not only visually accurate but also structurally representative of the real LNG tankers. These models served as the basis for both the wind tunnel experiments and the CFD simulations, providing a reliable platform for testing and validating the aerodynamic characteristics of different deck architectures. By constructing and utilizing these detailed 3D models, the study was able to conduct a thorough analysis of air resistance and its implications for LNG tanker design.

Table 1: Basic dimensions of the ship.

Dimension		**Unit**	
Length overall	*LOA*	[m]	249.555
Length between perpendicular	L_{PP}	[m]	237.0
Breadth	*B*	[m]	23.0
Draft	*T*	[m]	10.641
Deadweight	*DWT*	[t]	50,764
Tank capacity		[m^3]	87,603
Speed	*V*	[knots]	17.5

Two detailed 3D models of the LNG tanker were created using Rhinoceros software, as illustrated in Figures 1 and 2. These models were developed to a precise scale factor of 1/251, resulting in each model having an overall length of 1 meter. This specific scaling was chosen to balance accuracy with practicality, making the models suitable for wind tunnel testing while ensuring they remained manageable in size.

For the purpose of assessing wind resistance, the focus was placed exclusively on modelling the portion of the vessel that lies above the waterline. This decision was based on the understanding that the above-water structures primarily interact with the wind, thereby significantly influencing air resistance. Consequently, the underwater hull shape was not included in these models, as it does not contribute to the aerodynamic forces being studied.

In addition to this, the modelling process emphasized capturing the main parts of the superstructure, such as the cargo tanks, bridge, and other prominent features. Smaller details, which have a negligible impact on air resistance, were intentionally omitted to streamline the modelling process and reduce computational complexity. This approach allowed for a more efficient and focused analysis of the key elements affecting wind resistance.

By concentrating on the major structural components and simplifying the model to exclude minor details, the study ensured that the critical aerodynamic characteristics were accurately represented without unnecessary complications.

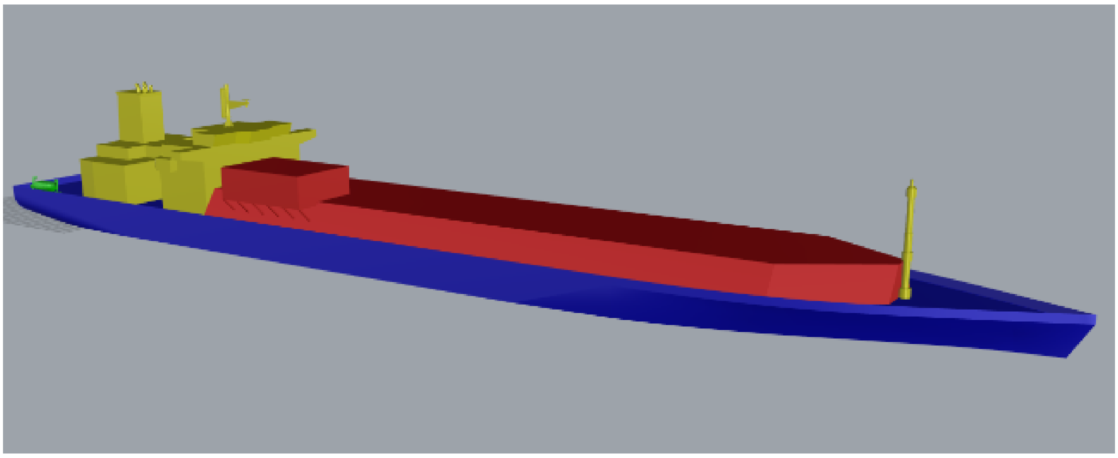


Figure 1: Ship model with rectangular tank

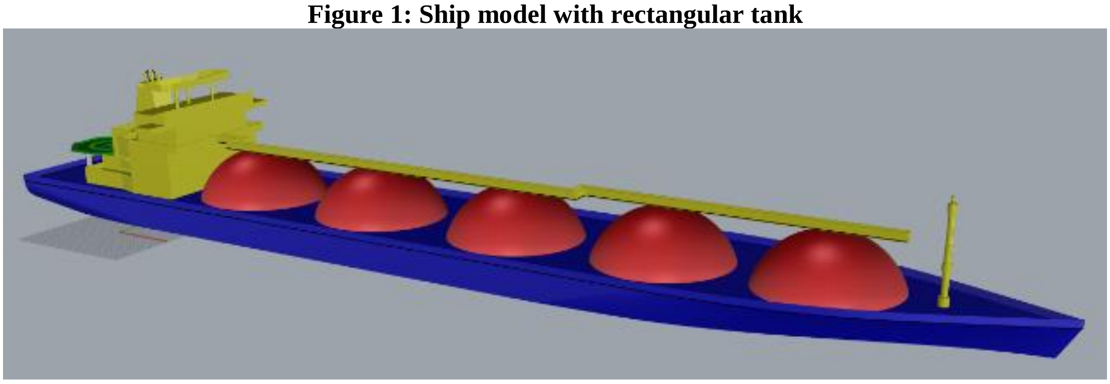


Figure 2: Ship model with sphere tank

The primary objective of the wind tunnel simulation is to determine the coefficient of air resistance for the LNG tanker models. To achieve this, each model is securely fixed at the base of the wind tunnel to ensure stability during testing. The models are then subjected to a series of rotations, with incremental adjustments of 10 degrees each time. At each 10-degree interval, precise measurements of the forces and torques acting on the model are recorded. Specifically, the tests are conducted using the two distinct models previously mentioned, with rotations ranging from 0 to 360 degrees. This comprehensive approach results in a total of 72 different test cases for each model.

During the testing process, three key quantities are meticulously measured and analyzed: the drag force in the x direction (Fx), the drift force in the y direction (Fy), and the torque around the z axis (Mz). These measurements are crucial for understanding the aerodynamic behavior of the models under varying wind conditions. To facilitate meaningful comparison and analysis, these measured quantities are converted into dimensionless coefficients. The conversion is performed using the following formulas, which normalize the forces and torque relative to the air density, wind velocity, and the lateral surface area of the models:

Drag coefficient:

$$C_{A(x,y)} = \frac{R_A}{\frac{1}{2}\rho_A V_A^2 P_A}$$

Torque coefficient

$$C_{MA} = \frac{M_A}{\frac{1}{2}\rho_A V_A^2 P_A L}$$

In which: R_A: Drag force (along X, Y axis)

M_A: Torque (around Z axis)

ρ_A : air density, $\rho_A = 1.2$ kg/m^3

V_A: wind velocity $V_A = 23.6$ m/s

P_A: lateral surface area.

$P_{A1} = 0.054627$ (m^2) – for rectangular tank

$P_{A2} = 0.0607687$ (m^2) – for spherical tank

L = 1 (m) – model length

Drag Coefficient (C_{Ax}): This coefficient represents the drag force normalized by the dynamic pressure and reference area, providing a non-dimensional measure of the aerodynamic resistance in the direction of the wind.

Drift Coefficient (C_{Ay}): Similar to the drag coefficient, this coefficient normalizes the drift force in the lateral direction, offering insights into the side forces experienced by the model.

Torque Coefficient (C_{MA}): This coefficient normalizes the torque around the z-axis, allowing for an assessment of the rotational effects of wind on the model.

By converting the measured forces and torques into these dimensionless coefficients, the study ensures that the results are comparable across different models and test conditions, providing a robust framework for analyzing the air resistance characteristics of the LNG tanker designs. This methodology not only enhances the accuracy of the results but also facilitates the application of the findings to real-world ship design and performance optimization.

2.2. CFD setup

The CFD calculations for this study are conducted using the software Star CCM+. This software is well-regarded for its robust capabilities in solving complex fluid dynamics problems. To ensure consistency and accuracy, the calculation domain is meticulously set to match the dimensions of the wind tunnel used in the experimental tests. This setup allows for a direct comparison between the CFD results and the empirical data obtained from wind tunnel experiments. The specific configuration of the calculation domain is illustrated in Figure 3.

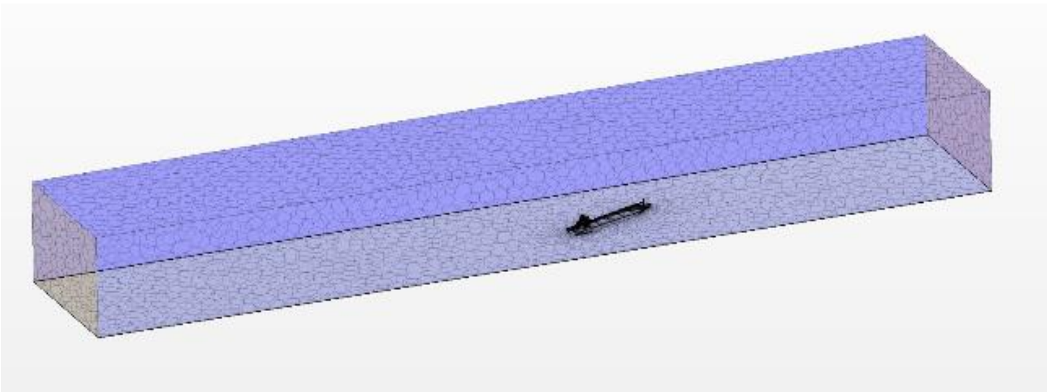


Figure 3: Calculation domain

Within this calculation domain, a polyhedral mesh is employed instead of the traditional hexahedral mesh. The choice of a polyhedral mesh is based on its superior performance in capturing complex geometrical features and providing more accurate simulation results. This type of mesh also offers better convergence properties and computational efficiency. For this study, approximately 550,000 grid cells are generated for each of the two LNG tanker models, ensuring a high-resolution representation of the flow field around the ship structures.

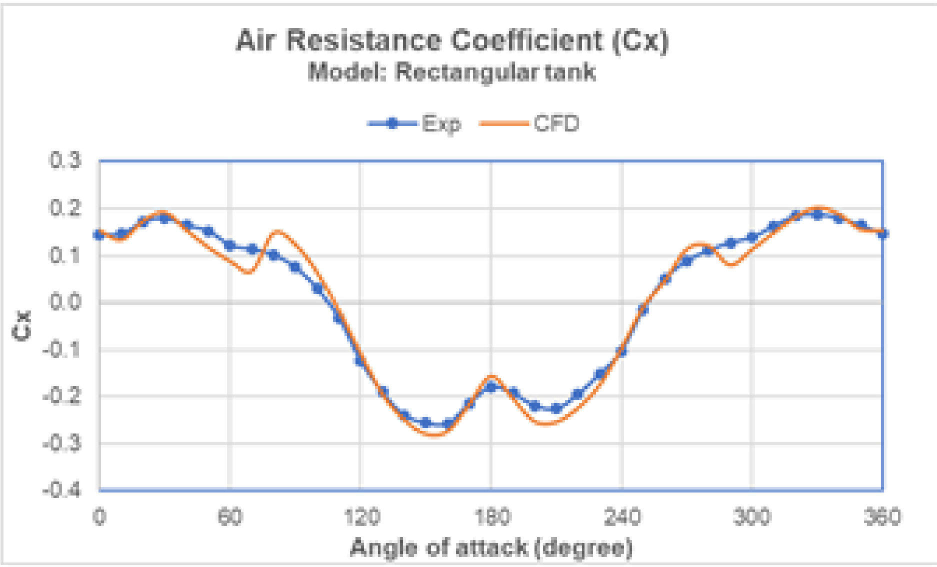
The boundary conditions for the CFD simulations are carefully defined to replicate the experimental setup. The wind velocity at the inlet boundary is set to match the conditions used in the wind tunnel tests, ensuring that the airflow entering the calculation domain is consistent with the experimental scenarios. The ship models themselves are treated as "wall boundaries," meaning that they are considered as solid surfaces interacting with the airflow. To accurately capture the turbulent flow characteristics, the k-ω SST turbulence model is utilized. This model is well-suited for predicting the complex boundary layer behaviour and separation phenomena that occur around the ship's superstructure. In this setup, the boundary layer is resolved directly using the "Low y+" setting, which ensures that the near-wall region is finely meshed and accurately simulated. This approach is crucial for capturing the detailed flow dynamics and ensuring that the results are reliable. Given the extensive number of calculations required—72 distinct cases in total—the author implemented an automated calculation method using macros in the Star CCM+ software. This practical approach significantly enhances the efficiency and manageability of the computational process. Instead of manually adjusting and recalculating for each rotational position of the model, the use of macros allows for a streamlined, automated sequence of operations.

For each rotation angle, the CFD calculation process is meticulously conducted over 1,000 iterations to ensure convergence and accuracy. After completing the iterations for a given angle, the macro automatically adjusts the model's orientation by rotating it 10 degrees. This automated rotation is followed by a re-gridding process, which updates the mesh to accurately reflect the new orientation of the model within the calculation domain. Once the re-gridding is complete, the macro proceeds to perform the next set of CFD calculations, continuing this cycle until all 72 rotational positions have been analysed. This method not only saves considerable time but also minimizes the potential for human error, ensuring that each calculation is performed consistently and systematically. The practical implementation of macros in Star CCM+ demonstrates a sophisticated approach to handling large-scale computational tasks. By automating the iterative and rotational processes, the study can efficiently generate a comprehensive dataset that captures the aerodynamic behaviour of the LNG tanker models across a full 360-degree range. This automated workflow highlights the practical benefits of integrating advanced computational techniques with meticulous planning and execution, ultimately leading to more accurate and reliable results.

In summary, the use of macros for automatic calculations in Star CCM+ represents a significant advancement in the practical application of CFD for aerodynamic analysis. This approach not only enhances the efficiency of the study but also ensures a high degree of precision, facilitating a thorough and detailed examination of air resistance characteristics in LNG tankers.

III. RESULT AND DISCUSSION

The results obtained from the CFD calculations, and the experimental tests are presented in Figures 4 and 5 below. These figures provide a detailed comparison of the air resistance characteristics as determined by both methods. Figure 4 illustrates the results for the model with the rectangular cargo tank, while Figure 5 presents the data for the model with the spherical cargo tank. Each figure includes plots of the drag coefficient, drift coefficient, and torque coefficient across the full range of rotational angles, from 0 to 360 degrees. By examining these results side-by-side, one can observe the consistency and discrepancies between the computational simulations and the empirical data. This comparison is crucial for validating the accuracy of the CFD method and understanding the aerodynamic performance of different tanker designs. The visual representation in these figures makes it easier to identify trends, anomalies, and key insights, providing a comprehensive overview of the study's findings.



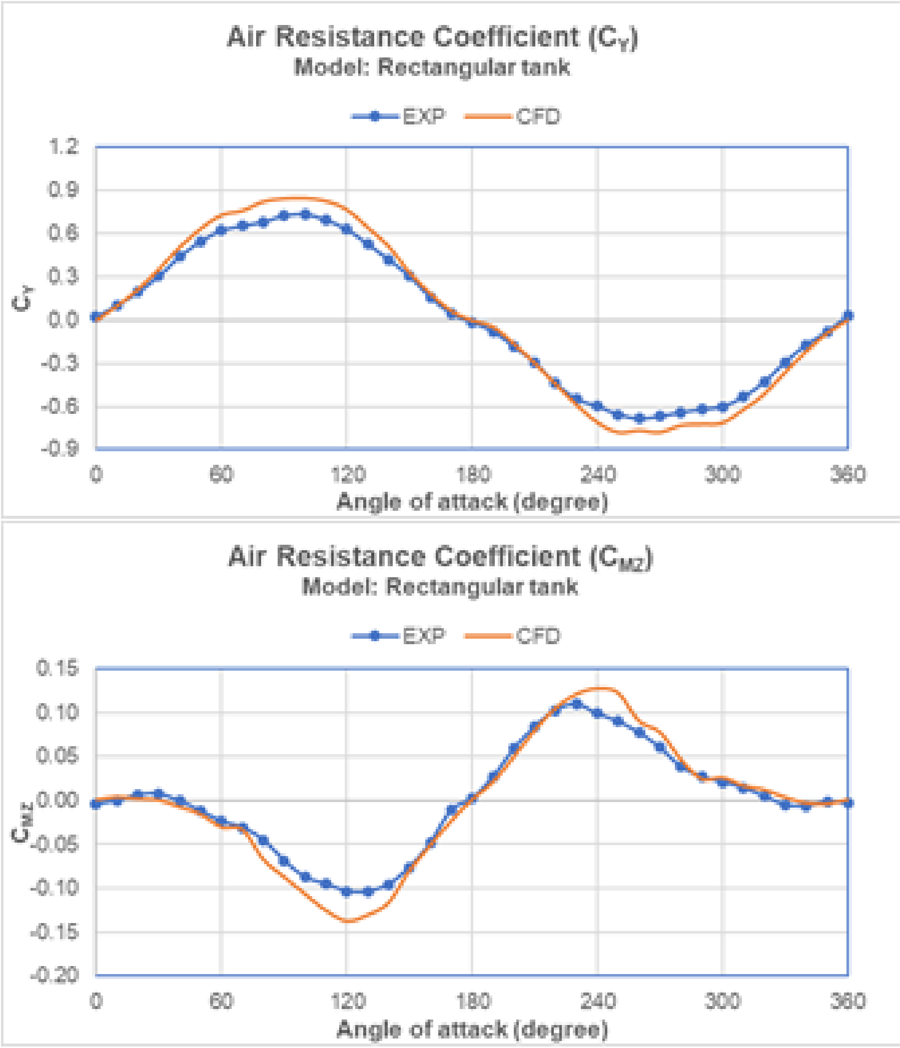
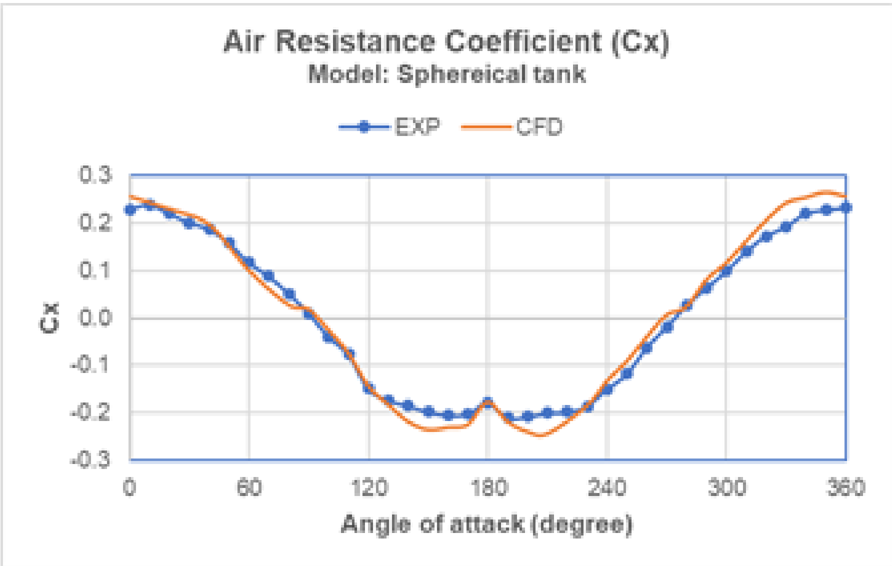
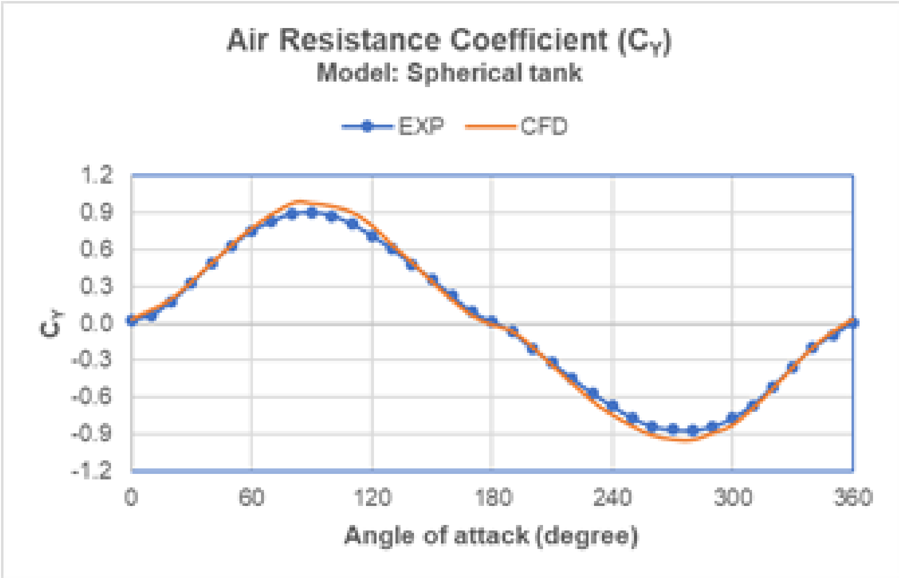


Figure 4: Air resistance coefficient – rectangular tank





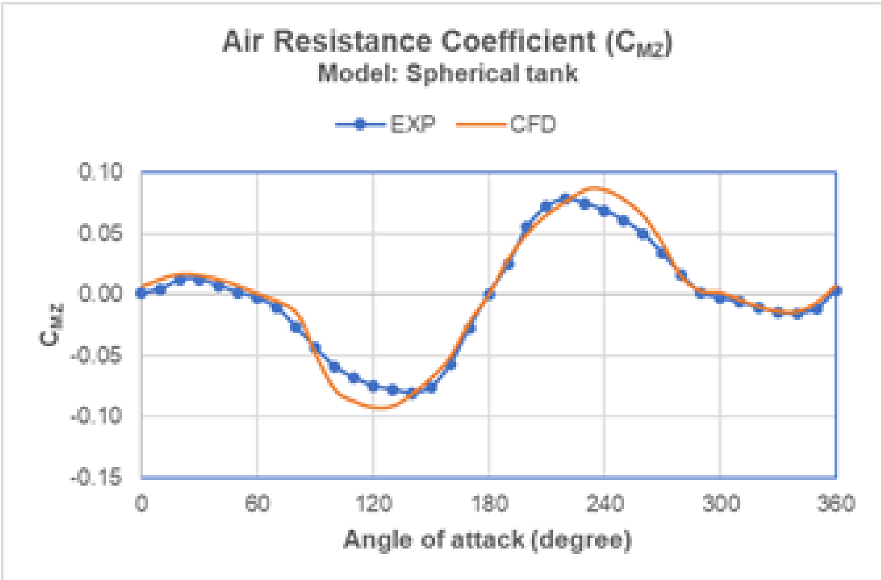


Figure 5: Air resistance coefficient – spherical tank

A detailed comparison of the drag coefficients of the two models reveals significant differences in their aerodynamic performance. Notably, the model with the spherical cargo tank exhibits a torque coefficient (CMZ about the Z axis) that is approximately 30% smaller than that of the model with the rectangular cargo tank. This substantial reduction in torque is particularly advantageous because high torque can significantly impair a ship's maneuverability, especially when navigating through confined spaces such as canals and wharf areas where precise control is critical. Consequently, the spherical tank design offers a clear advantage in terms of reducing torque-induced challenges, enhancing the vessel's handling capabilities in such scenarios.

In addition to the torque differences, the analysis shows that the drag coefficients in the x direction (Cx) and the y direction (Cy) for both tank types are quite similar. This similarity indicates that, in terms of direct aerodynamic drag, both designs perform comparably. However, the advantage of the spherical tank's reduced torque remains a significant factor favoring its overall aerodynamic efficiency and operational performance. The CFD calculation results closely align with the experimental data, underscoring the accuracy and reliability of the CFD methodology employed in this study. This is particularly evident in the drag coefficient Cy, which is the largest among the three coefficients (Cx, Cy, and CMZ). The strong correlation between the CFD results and the wind tunnel test data validates the computational approach and demonstrates its effectiveness in predicting the aerodynamic behavior of the LNG tanker models. The graphical representation of the results further reinforces the reliability of the CFD simulations. The trends observed in the graphs are consistent with the empirical data, confirming that the CFD method can accurately capture the complex interactions of air flows around the ship's superstructure. Although some discrepancies may exist at specific angles, the overall trend indicates that the CFD results are robust and can be trusted for engineering applications.

IV. CONCLUSION

The close agreement between the CFD results and the experimental data obtained from wind tunnel tests validates the use of Computational Fluid Dynamics as a reliable and effective tool for aerodynamic analysis in ship design. This validation is significant because it demonstrates that CFD can accurately replicate real-world conditions and provide detailed insights into the aerodynamic performance of large maritime vessels, such as LNG tankers.

One of the key strengths of this study is its practical approach to integrating CFD with macro implementing. By employing an automated calculation method through macros in Star CCM+, the study efficiently handled a large number of simulation cases, streamlining the process and reducing the computational load. This practical methodology not only saves time but also enhances the precision and consistency of the results, making it a feasible option for extensive aerodynamic studies.

REFERENCES

[1]. W.D. Janssen, B. Blocken, H.J. van Wijhe, CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications, Journal of Wind Engineering and Industrial Aerodynamics, Volume 166, 2017, Pages 106-116, ISSN 0167-6105,

[2]. W.D. Janssen, B. Blocken, H.J. van Wijhe, CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications, Journal of Wind Engineering and Industrial Aerodynamics, Volume 166, 2017, Pages 106-116, ISSN 0167-6105,

[3]. Ouchi, K, et. al. (2014). "A Study on Air Drag Reduction on the Large Container Ship in the Sea," International conference Design & Operation of Container Ships, London, pp 107-114.

[4]. Kim, Y, et. al. (2015). "Design and Performance Evaluation of Superstructure Modification for Air Drag Reduction of a Container Ship," Proc 25th International Ocean and Polar Engineering Conference, Hawaii, Vol 4, pp 894-901.

[5]. Bertram, V.,(2011) Practical ship hydrodynamics, Elsevier.

[6]. Artjushkov, L., (1968) Wall effect correction for shallow water model tests. NE Coast Institution of Engineers and Shipbuilders.

[7]. Geerts, S., Verwerft, B., Vantorre, M., and Van Rompuy, F., (2010) Improving the efficiency of small inland vessels. Proc., 7th European Inland Waterway Navigation Conf., Budapest Univ. of Technology and Economics, Budapest, Hungary.,

[8]. Prakash, S. and B. Chandra, (2013) Numerical estimation of shallow water resistance of a river-sea ship using CFD. International journal of computer applications, **71**(5).

[9]. Pacuraru, F. and L. Domnisoru (2017). Numerical investigation of shallow water effect on a barge ship resistance. in IOP Conference Series: Materials Science and Engineering. IOP Publishing.