A STUDY OF HULL RESISTENCE AND PRESSURE DISTRIBUTION OF CONTAINER MOVING ON FRESHWATER AND SALTWATER

BY

OBANOR GIFT OSAYONAMEN
OMOIKE ELIJAH ONOSHIOAGBE
ADE-AJAYI DANIEL OLUWAFUNMIBI

SUPERVISED BY

PROF. A. I. OBANOR

DEPARTMENT OF MECHANICAL ENGINNERING
MARINE ENGINERING PROGRAM
FACULTY OF ENGINEERING, UNIVERSITY OF BENIN
BENIN CITY, EDO STATE
NIGERIA

NOVEMBER 2025

DECLARATION

We, OBANOR GIFT OSAYONAMEN, OMOIKE ELIJAH ONOSHIOAGBE and ADE-AJAYI DANIEL OLUWAFUNMIBI, hereby declare that this project work titled "**A STUDY OF HULL RESISTENCE AND PRESSURE DISTRIBUTION OF CONTAINER MOVING ON FRESHWATER AND SALTWATER"** is an original work carried out out by us in the Depertment Of Mechnical Engineering, Marine Engineering Program, University Of Benin, under the supervision of Professor A. I. Obanor.

OBANOR GIFT OSAYONAMEN ____________________________
ENG2006350 SIGNATURE AND DATE

OMOIKE ELIJAH ONOSHIOAGBE ____________________________
ENG20106263 SIGNATURE AND DATE

ADE-AJAYI DANIEL OLUWAFUNMIBI ____________________________
ENG2010799 SIGNATURE AND DATE

CERTIFICATION

This is to certify that this project title is "**A STUDY OF HULL RESISTENCE AND PRESSURE DISTRIBUTION OF CONTAINER MOVING ON FRESHWATER AND SALTWATER"** was carried out by OBANOR GIFT OSAYONAMEN, OMOIKE ELIJAH ONOSHIOAGBE and ADE-AJAYI DANIEL OLUWAFUNMIBI with the matriculation number ENG2006350, ENG20106263 and ENG2010799 respectively in the Department Of Mechanical Engineering, Marine Engineering Program, University Of Benin, Benin City, Edo State.

Prof. A. I. Obanor — Date
Project Supervisor

Engr. W. Jaja — Date
Project Coordinator

Engr. Prof. Ebunilo — Date
Head of Department

ACKNOWLEDGEMENT

Firstly, we express my sincere gratitude to the Almighty God, the Creator of all creatures for the Grace, Love and Mercy throughout this endeavor. All Glory and Honor be unto Him forever.

We also appreciate our supervisor, Prof. A. I. Obanor for his guidance, support and feedback throughout this project.

We would like to appreciate our parent and our family members thank you all for the patience, understanding and sacrifice made in ensuring our academic success.

Finally, we want to express my heartfelt thanks to the department, our course mate and friends.

DEDICATION

This project is dedicated to the almighty GOD.

ABSTRACT

This study investigates the hull resistance and pressure distribution of a container vessel operating in both freshwater and saltwater environments using Computational Fluid Dynamics (CFD) simulation. The analysis employs the Reynolds-Averaged Navier–Stokes (RANS) equations with the k–ω SST turbulence model and the Volume of Fluid (VOF) method to accurately capture hydrodynamic behavior around the hull. Results reveal that total resistance increases quadratically with speed in both environments but is consistently higher in saltwater due to its greater density and viscosity, which amplify dynamic pressure and wave-making effects. Pressure contour analysis shows high-pressure zones at the bow and low-pressure regions near the stern, influencing overall resistance characteristics. The findings emphasize the significance of environmental conditions on vessel performance, offering valuable insights for hull form optimization, propulsion efficiency, and energy-saving ship design.

TABLE OF CONTENT

Title page ……………………………………………………………….
Declaration ……………………...…………………………….. i
Certification ….…………………………………………………… ii
Acknowledgement …………………...………………………… iii
Dedication …………………………..…………………………….. iv
Abstract ……………………………………………………………….. v
Table of Content ……………………..………………………….. vi
List of Tables ……………………………………………………. vii
List of Equations ………………………………………………… ix
List of Graphs …………………………………………………….. x
List of Figures ……………………………………………………. xi

CHAPTER ONE – INTRODUCTION
1.0 Introduction……………………………………………………..... 1
1.1 Background of the Study………………..…………………… . 1
1.2 Statement of the Problem……………………….…………….. 2
1.3 Aim and Objectives of the Study………………..……………….. 3
1.4 Research Questions…………………………………………… 3
1.5 Significance of the Study……………………………………… 4
1.6 Scope of the Study………………………………………………... 6

CHAPTER TWO – LITERATURE REVIEW
2.1 Introduction……………………………….………………….. 7
2.2 Definition of Terms…………………………………………….. 7
2.3 Overview of Ship Hydrodynamics………………………… ….. 10
2.4 Pressure Distribution on Ship Hulls…………………………… 11
2.5 Components of Hull Resistance……………………………….. 12
2.6 The Rise of Computational Fluid Dynamics (CFD) in Ship Hydrodynamics ……..…………………………………………….. 16
2.7 Hull Resistance and Pressure Distribution Impact on Container Vessel………………… ……………………………………………… 18
2.7.2 Decomposing and Quantifying Resistance Components ………. 18
2.8 Factors Affecting Hull Resistance and Pressure Distribution…….. 18
2.9 Gaps in Current Research and Project Justification……………… 19

CHAPTER THREE – RESEARCH METHODOLOGY
3.1 Introduction……………………………………………………. 22
3.2 System Details in Running the Simulation……………………….. 22
3.3 Hull Geometry and Model Preparation………………………….. 23

3.4 Physical Models and Governing Equations……………………… 24
3.4.1 Turbulence Model (k–ω SST)………………………………. 25
3.4.2 Free Surface and Wave Modelling………………………….. 25
3.4.3 Resistance Decomposition…………………………………… 26
3.4.4 Fluid Properties: Saltwater and Freshwater………………….. 26
3.5 Boundary and Operating Conditions……………………………… 27
3.6 Input Parameters…………………………………………………... 28
3.7 Post-Processing and Data Extraction…………………………….. 32
3.8 Validation…………………………………………………………. 34

CHAPTER FOUR – RESULTS AND DISCUSSION

4.1 Introduction………………………………………………………. 35
4.2 Effect of Speed on Resistance in Freshwater…………………….. 35
4.3 Effect of Speed on Resistance in Saltwater………………………. 37
4.4 Discussion on the Effect of Speed on Resistance in Saltwater and Freshwater……………………………………………………………… 39
4.5 Pressure Distribution Contours…………………………………… 40
4.5.1 In Freshwater…………………………………………………… 41
4.5.2 In Saltwater…………………………………………………….. 41

CHAPTER FIVE – CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion………………………………………………………… 43
5.2 Recommendations………………………………………………… 43

Reference ……………………………………………………………...45

LIST OF TABLES

Table 2.1 Differences Between Freshwater and Saltwater (NOAA, 2024)

Table 3.1 Basic System Details in Running the Simulation

Table 3.2 Computational Domain

Table 3.3 Basic Mesh Dimensions

Table 3.4 Ambient Conditions

Table 4.1 Resistance vs Speed Result in Freshwater

Table 4.2 Resistance vs Speed Result in Saltwater

LIST OF EQUATIONS

Equation 2.1 Total Resistance components

Equation 3.1 RANS Momentum Equation

Equation 3.2 Total Ship Resistance

Equation 4.1 Freshwater resistance fit

Equation 4.2 Correlation coefficient

Equation 4.3 Saltwater resistance polynomial fit

LIST OF GRAPHS

Graph 4.1 Resistance Against Speed in Freshwater

Graph 4.2 Resistance Against Speed in Saltwater

LIST OF FIGURES

Figure 3.1 Hull Geometry Used

Figure 3.2 Velocity Contours Create Wake

Figure 4.1 Pressure Contours in Freshwater

Figure 4.2 Pressure Contours in Saltwater

CHAPTER ONE

1.0 INTRODUCTION:

1.1 Background of the Study

Pressure distribution describes the variation of water pressure across a ship's hull as it moves through the water. Understanding pressure distribution helps in identifying areas of high pressure, which can lead to increased resistance and potential structural issues(Faltinsen, 2005; Larsson & Raven, 2010). Hull shape (e.g., bulbous bow, bilge keels), speed, and wave conditions all influence pressure distribution. A low-pressure region often forms upstream of the flow convergence line near the keel, Bilge keels can create both positive and negative pressure zones, affecting roll damping. Roughness on the hull (e.g., from biofouling) can alter pressure distribution and increase resistance(Schultz, 2007).

Hull resistance, or drag, is the force that opposes a ship's motion through the water. Total resistance is typically broken down into frictional resistance, wave-making resistance, and eddy-making resistance (ITTC, 2017). Lowering resistance is essential for fuel efficiency and speed. Different vessel hull shapes significantly affect how ships interact with water during motion influencing fuel efficiency, wave-making resistance, maneuverability, and stability. In commercial shipping, naval operations, and passenger ferry design, selecting or modifying the wrong hull geometry can lead to excessive hydrodynamic resistance, inefficient propulsion, and increased emissions (IMO, 2020).

Due to the growing push toward greener marine transport and cost optimization, there is a critical need to understand which hull forms perform best under different operational profiles especially for mid-sized vessels like coastal tankers, ferries, or fast supply boats. Pressure distribution and hull resistance helps ship designers and naval architects choose better hull forms. This improve fuel efficiency, supports environmental goals by reducing fuel usage and carbon emissions in coastal shipping, reducing trial and error in physical prototyping (Tezdogan, Incecik, & Turan, 2015).

.

The pressure distribution around a ship hull is complex, varying significantly due to the ship's motion, hull shape, and surrounding environment like waves and shallow water. It's characterized by regions of high and low pressure that influence the ship's resistance, stability, and overall performance. As a ship moves, the pressure on the hull deviates from static pressure due to the dynamic effects of the water flow (White, 2016).

1.2. Statement of the Problem

Different vessel hull shapes significantly affect how ships interact with water during motion influencing fuel efficiency, wave-making resistance, maneuverability, and stability. In commercial shipping, naval operations and passenger ferry design, selecting or modifying the wrong hull geometry can lead to excessive hydrodynamic resistance, inefficient propulsion, and increased emissions.

Due to the growing push toward greener marine transport and cost optimization, there is a critical need to understand which hull forms perform best under different operational profiles especially for mid-sized vessels like coastal tankers, ferries, or fast supply boats. In the design and operation of sea-going vessels, accurate prediction of hydrodynamic resistance is essential for improving fuel efficiency, optimizing hull forms, and ensuring operational safety. A significant component of this resistance arises from pressure distribution over the hull surface, especially during varying sea conditions and vessel speeds. Traditional experimental methods like towing tank tests, though accurate, are time-consuming and costly. In contrast, numerical methods such as Computational Fluid Dynamics (CFD) offer an efficient alternative for investigating flow behavior around hulls.

However, challenges remain in achieving high-fidelity simulations that accurately capture complex flow phenomena, including turbulence, wave-making, boundary layer effects, and viscous resistance components. Furthermore, limited understanding of pressure distribution patterns leads to sub-optimal hull designs and excessive energy consumption. Therefore, a need exists to numerically analyze and understand pressure distribution and its correlation to total hull resistance for various hull forms and operational conditions.

1.3 AIM AND OBJECTIVES OF THE STUDY

1.3.1 Aim

The aim of the study is to analyse the hull resistance and pressure distribution of container vessel moving in freshwater and saltwater.

1.3.2 Objectives of the Study

The to achieve the aim the objectives are

1. To study the software (Computation Fluid Dynamics. Solidwork).
2. The application of the solidwork to determine the hull resistance and pressure distribution of the container vessel moving in freshwater and saltwater.
3. To compare the characteristic of the hull resistance in freshwater and saltwater in order to achieve hydrodynamic efficiency.
4. To study pressure distribution along the ship hull in order to achieve structural integrity of the hull.

1.4 RESEARCH QUESTIONS

1. How does pressure distribution vary along the hull surface of container vessel under different speeds and sea conditions?
2. What is the relationship between pressure distribution and the total hydrodynamic resistance experienced by the vessel?

3. Can Computational Fluid Dynamics (CFD) methods accurately predict pressure distribution and hull resistance compared to experimental or empirical approaches?
4. How do changes in hull geometry (e.g., bulbous bow, stern shape, length-to-beam ratio) affect the pressure distribution and total resistance?

1.5 SIGNIFICANCE OF THE STUDY

1. Contribution to Ship Design Optimization

This project provides critical insights into pressure distribution along a ship's hull, which is directly linked to hydrodynamic resistance. Accurate prediction of this resistance is essential for designing more efficient hull forms (Molland et al., 2011). The numerical approach allows designers to identify high-pressure zones and flow separation areas early in the design phase, leading to optimized shapes that reduce drag and fuel consumption.

2. Enhancement of Marine Energy Efficiency

With increasing global pressure to reduce greenhouse gas emissions from ships (e.g., IMO's Energy Efficiency Design Index - EEDI), this study aligns with international standards. Minimizing resistance leads to lower engine loads, reducing fuel consumption and emissions, and contributing to sustainability and compliance with environmental regulations.

3. Advancement of Computational Fluid Dynamics (CFD) Application

The project validates the effectiveness of CFD tools in predicting flow behavior and resistance components with high accuracy. It demonstrates that CFD can serve as a reliable alternative or supplement to physical model tests, supporting its wider acceptance in industry and academia as a standardized design and analysis method.

4. Cost and Time Efficiency for the Marine Industry

Numerical investigations significantly reduce the cost and time associated with repeated physical testing in towing tanks. By applying validated CFD methods, shipbuilders and naval architects can perform multiple design iterations efficiently, accelerating the development cycle and improving competitiveness.

5. Contribution to Marine Engineering Research and Standards

This research supports the development of standardized numerical methods and validation protocols for resistance prediction. It contributes to the growing body of academic and industrial literature on ship hydrodynamics and pressure-flow interaction, enhancing future benchmarking and best practices.

6. Structural Safety and Performance

Understanding pressure distribution is not only vital for performance but also for structural integrity. Over-pressurized hull sections can lead to fatigue or failure. This project aids in predicting such risks and improving structural design accordingly, ensuring the safety of sea-going vessels.

The study of hull resistance and pressure distribution of container vessel moving in freshwater and saltwater is significant because it helps optimize hull design for improved performance, fuel efficiency and safety. By accurately predicting pressure distribution and resistance, engineers can minimize drag, reduce fuel consumption, and enhance a vessel's maneuverability and stability in different sea conditions. This research also aids in understanding the complex hydrodynamic interactions between the hull and the surrounding water, particularly in shallow or restricted waters, which is crucial for preventing grounding and ensuring safe navigation.

1.6 Scope of the Study

The scope of this study focuses on analyzing and comparing the hull resistance and pressure distribution of a container vessel operating in freshwater and saltwater environments. The research will employ Computational Fluid Dynamics (CFD) techniques to simulate the flow of water around the hull and determine how variations in water density and viscosity between the two environments influence the vessel's hydrodynamic performance. The study will model a standard container vessel hull under steady-state conditions and examine how speed variations affect total resistance and surface pressure distribution. Parameters such as Froude number, Reynolds number, and boundary layer behavior will be considered to capture realistic flow characteristics and resistance components.

The investigation will be limited to calm, deep-water conditions, neglecting the influence of waves, wind, and hull roughness. Both freshwater with density of 1000 kg/m^3 and saltwater with density of 1025 kg/m^3 conditions will be simulated using the same hull geometry and operational speeds for consistent comparison. Results will include pressure contour and resistance-speed curves to highlight performance differences between the two fluids. The study's outcomes aim to improve understanding of how environmental conditions impact hull hydrodynamics, offering insights that can support hull form optimization and more accurate resistance predictions in different marine environments.

CHAPTER TWO

LITRATURE REVIEW

2.1 Introduction

The hydrodynamic performance of a sea-going vessel depends largely on the interaction between the hull and the surrounding fluid medium. Pressure distribution and hull resistance are critical aspects of ship design that determine propulsion power requirements, fuel consumption,

and overall efficiency. Over the years, several researchers and organizations have developed theoretical, empirical, and computational models to analyze and predict these phenomena. This chapter reviews existing literature, theories, and models related to pressure distribution, hull resistance, and their implications in marine engineering. (Molland, Turnock, & Hudson, 2011).

2.2 Definition Of Terms

1. Pressure Distribution

The variation of pressure along the surface of the ship's hull caused by water flow. It affects both the resistance experienced by the vessel and the structural loads on the hull. (Larsson & Raven, 2010).

1. Hull Resistance

The total force that opposes the forward motion of a ship through water. It consists of components such as frictional resistance, pressure (form) resistance, and wave-making resistance. (ITTC, 2017)

2. Container Vessel

A container ship is a cargo ship that carries all of its load in truck-size container, in a technique called containerization. Container ship are common means of commercial transport and now carry most seagoing non-bulk cargo. Container ship capacity is measured in twenty-foot equivalent units (TEU). Typical loads are a mix of 20-foot (1-TEU) and 40-foot (2-TEU) International Organization for Standardization (ISO) for containers, with the latter predominant (Wikipedia). Examples of container ship are Maersk Triple-E class, Madrid Maersk, Xin Los Angeles and more.

3. Freshwater

Freshwater is a body of water, such as a river, lake, or canal, characterized by a low concentration of dissolved salts, typically less than 0.5 to 1 parts per thousand (ppt). It is essentially non-saline, in contrast to the average salinity of seawater, which is about 35parts per thousand.

Freshwater can also be referred to as any naturally occurring liquid or frozen water containing low concentrations of dissolved salts and other total dissolved solids. Freshwater may encompass frozen and melt water in ice sheets, ice caps, glaciers, snowfields and icebergs, natural precipitations such as rainfall, snowfall, hail/sleet and graupel, and surface runoffs that form inland bodies of water such as wetlands, ponds, lakes, rivers, streams, as well as groundwater contained in aquifers, subterranean rivers and lakes.

1. Saltwater

Saltwater is defined as water containing a significant concentration of dissolved salts, with the most common example being the water in Earth's oceans and seas (seawater). The average salinity of ocean water is approximately 35 parts per thousand (ppt), meaning that for every kilogram of seawater, about 35 grams are dissolved solids, predominantly sodium chloride i.e table salt. This high salt content gives saltwater a higher density than freshwater and makes it unsuitable for drinking or most agricultural uses without desalination, yet it is essential for marine ecosystems, supporting a vast array of life.

Table 2.1 Differences Between Freshwater And Saltwater (NOAA, 2024)

S/N	Parameter	Saltwater	Freshwater
1	Salinity	High salinity (about 35 parts per thousand)	Very low salinity (less than 0.5 parts per thousand)
2	Density	Denser due to dissolved salt (1025 kg/m^3)	Less dense (1000 kg/m^3)
3	Freezing Point	Lower freezing point (about -2 degree celsius)	Freezing point at 0 degree celsius
4	Electrical Conductivity	High conductivity because of ions	Low conductivty due to fewer ions
5	Bouyancy	Greater buoyancy (object float easily)	Less bouyancy
6	Taste	Salty	Tasteless
7	Source	Found in oceans and seas	Found in rivers lakes and streams
8	Marine life	Contain salt-tolerant organisms	Contain freshwater organisms
9	Corrosiveness	More corrosive to metals and material	Less corrosive
10	Evaporation Residue	Leaves salt crystals after evaporation	Leave little or no residue after evaporation

2. Hull Form

The shape and geometry of the submerged part of a vessel. It influences the flow of water around the ship and thus directly affects pressure distribution and resistance. (SNAME, 1988).

3. Hydrodynamics

The study of fluids (liquids) in motion. In marine engineering, it refers to how water interacts with the ship's hull during movement. (Faltinsen, 2005).

2.3 Overview of Ship Hydrodynamics

Ship hydrodynamics is the study of how water and air flow around a vessel's hull as it moves. It involves understanding the physical behavior of viscous and turbulent flow, boundary layer development, wake formation, and wave-making. (Molland et al. 2011). The forces acting on the hull are divided into components that contribute to resistance and propulsion.

The total resistance of a vessel is the addition of all hydrodynamic forces opposing its motion. The flow pattern around the hull determines both the pressure field and the energy losses through viscous and wave effects. As the ship's speed increases, the interaction between viscous and wave-making effects becomes more significant, influencing the total power required for propulsion.

The economic viability and environmental impact of a ship are largely dictated by its hull resistance, the force opposing its motion through water. A comprehensive understanding of this force is paramount in naval architecture and marine engineering.

Historically, the primary method for quantifying these forces was through towing tank experiments using scaled ship models. While highly accurate, this method is labor-intensive, expensive, and limited in its ability to provide granular, real-time data on the intricate flow dynamics around the hull, particularly the pressure distribution. The detailed visualization of pressure fields, which is crucial for optimizing hull forms, was not feasible with this approach.

2.4 Pressure Distribution on Ship Hulls

Pressure distribution refers to the variation of static and dynamic pressure along the surface of the hull as water flows past it. The flow at the bow region is generally characterized by high pressure zones due to flow stagnation, while the mid-ship and stern regions experience low-pressure zones due to flow acceleration and separation.

Accurate prediction of pressure distribution is vital for determining both resistance and structural loads. High-pressure areas contribute to form resistance and may also affect ship stability and vibration. Bernoulli's principle relates pressure and velocity in fluid flow which state that an increase in the speed of a fluid occurs simultaneously with a decrease in pressure or decrease in the fluid's potential energy, and together with the Navier–Stokes equations, provides a mathematical framework for analyzing pressure variations.

The flow separation, turbulence, and boundary layer effects strongly influence pressure distribution. Computational Fluid Dynamics (CFD) has become an essential tool for visualizing these variations and understanding their impact on performance. (Molland et al., 2011).

2.5 Components of Hull Resistance

The total hydrodynamic resistance of a ship is made up of several hydrodynamic forces acting opposite to its motion through water. The interaction between the hull and the surrounding fluid causes the forces.

According to Faltinsen (2005), total resistance can be expressed as the sum of frictional resistance, wave-making resistance and viscous pressure (form) resistance. In calm water, these are often grouped as calm-water resistance, while other minor component such as air resistance and appendage resistance are added to obtain the total effective resistance.

The International Towing Tank Conference (ITTC) conventionally divides total resistance ((RT)) into:

$$RT = RF + RW + RE + RA \text{ ---------- equation 2.1}$$

where:

RF = Frictional Resistance

RW = Wave-making Resistance

RE = Eddy or Form Resistance

RA = Air Resistance (ITTC, 2017)

1. Frictional Resistance

This arises due to the viscosity of water as the ship moves through it experiencing a drag force due to the friction between the ship's hull and the water. A boundary layer forms along the hull surface, where velocity gradients produce shear stresses that oppose motion. The ITTC 1957 formula is widely used to calculate frictional resistance coefficients based on the Reynolds number. According to park et al., 2020, hull coating and surface treatment can reduce frictional resistance and improve propulsion efficiency.

2. Wave-making Resistance

As a ship moves on the free surface, it creates a system of waves that require energy to generate and maintain. This energy loss appears as wave-making resistance. Michell's thin ship theory (1898) provides the first theoretical model to estimate this resistance by simplying the hull to a thin central plane. Container vessels operating at high service speed are more susceptible to wave-making resistance; hence, optimization of the bulbous bow and hull form design is essential to minimize energy losses (Kapsenberg, 2011).

3. Eddy or Form Resistance

Eddy resistance caused by separation of the water flow around the hull and the creation of vortices at the stern and abrupt hull shapes. It depends on hull shape, appendages, and flow smoothness. Streamlined hull forms help reduce this component.

4. Air Resistance

This is caused by wind and air flow around the freeboard and superstructure. This is usually small in percentage of the total resistance in slow speed vessels or calm conditions and it can become significant for high-speed vessels or ships with large exposed areas. The effect of air resistance determine the hdrodynamic performance of a container vessel and computational fluid dynamics simulation are now widely used to quantify and optimize itr for modern ship designs (Sahoo et al 2021).

Mitchell's thin ship theory (1898) provides the first analytical model to estimate wave-making resistance. Frictional resistance is typically calculated using the ITTC 1957 correlation line, based on the Reynolds number. Eddy and air resistances are influenced by hull form and superstructure design (Larsson & Raven, 2010).

2.6 The Rise of Computational Fluid Dynamics (CFD) in Ship Hydrodynamics

The advancement of Computational Fluid Dynamics (CFD) has revolutionized the field of ship hydrodynamics by providing a detailed, cost-effective and efficient method for analyzing fluid flow around ship hulls. Traditionally, naval architects relied on towing tank experiments and empirical formulas to predict ship resistance, propulsion, and sea-keeping characteristics. However, these methods are often time-consuming, expensive, and limited in scope. With the emergence of high-performance computing and advanced numerical algorithms, CFD has become a powerful alternative for simulating the complex flow phenomena associated with marine vessels (Larsson & Stern, 2010). The technique enables engineers to study detailed pressure distribution, wake flow, resistance and free-surface effects that were previously difficult to capture experimentally (Faltinsen, 2005).

CFD applications in ship hydrodynamics have expanded significantly since the 1990s, driven by improvements in turbulence modeling and grid generation techniques. The Reynolds-Averaged Navier–Stokes (RANS) equations have been widely used to predict viscous flow and total resistance with reasonable accuracy (Hino, 2005). Later, hybrid models such as Detached Eddy Simulation (DES) and Large Eddy Simulation (LES) further improved the prediction of flow separation and vortex shedding around hulls. Researchers such as Stern et al. (2013) demonstrated that CFD could accurately reproduce towing tank results for hull resistance, propeller–hull interaction, and pressure distribution, leading to its acceptance as a complementary tool in both academia and industry.

One major advantage of CFD is its ability to perform parametric and optimization studies without requiring physical prototypes. Using numerical simulations, engineers can test multiple hull designs, bulbous bow shapes, or appendage configurations to minimize resistance and improve fuel efficiency (Kouh, Kim, & Lee, 2020). This approach significantly reduces design costs and development time. For instance, modern shipyards now use CFD-based optimization to refine hull forms for both freshwater and saltwater operations, ensuring better performance under varying environmental conditions (Sahoo et al., 2021). Moreover, CFD has proven valuable in predicting green ship technologies, such as air lubrication systems and energy-saving devices.

In recent years, CFD has also been integrated with Computational Structural Mechanics (CSM) and Computational Heat Transfer (CHT) for a more holistic understanding of ship performance. Multi-physics simulations now allow for the coupled analysis of hydrodynamic loads, structural deformation, and thermal effects (Kumar & Kim, 2021). This multidisciplinary approach is particularly important for high-speed vessels, offshore platforms, and container ships operating under diverse sea states. The accuracy and versatility of CFD have thus made it indispensable for modern marine hydrodynamic research, ship design, and performance assessment.

Despite its benefits, challenges remain in CFD applications, including computational cost, turbulence model limitations, and the need for experimental validation. Nevertheless, continuous progress in high-performance computing and open-source CFD software, such as OpenFOAM, has made simulation tools more accessible (Ferziger et al., 2020). Consequently, CFD continues to rise as a critical tool in ship hydrodynamics bridging the gap between theory, experiment, and practical ship design. Its integration into the early design stages ensures safer, faster, and more energy-efficient vessels that meet the evolving demands of the maritime industry.

2.7 Hull Resistance And Pressure Distribution Impact On Container Vessel

The pressure distribution throughout a container vessel's hull is critical in determining its hydrodynamic performance and total resistance. As a ship passes through water, the flow interacts with the hull surface, causing pressure differences due to variations in velocity and flow separation. According to Faltinsen (2005), flow stagnation causes high pressure in the bow, while low pressure zones arise along the hull sides and stern, influencing the lift and drag forces acting on the vessel. The resulting pressure field defines the form resistance (or pressure resistance), which is caused by the integrated pressure differential over the hull surface. Proper understanding of pressure distribution is therefore critical in designing hull geometry to reduce resistance and improve propulsion efficiency.

Frictional and wave-making resistance are two aspects of hull resistance in container ships that are directly impacted by pressure distribution. Higher drag forces and energy losses result from flow separation and turbulence caused by an uneven or poorly regulated pressure field through hull design. Streamlined hull forms with smooth pressure gradients minimize flow separation at the stern, lowering both viscous and wave-making resistance, according to studies by Kim et al. (2018) and Sahoo et al. (2021). On the other hand, sudden changes in the hull's curvature or a poorly designed bulbous bow may result in localized pressure decreases, which can lead to the production of vortices and an increase in overall resistance. In order to analyze these pressure patterns and optimize hull forms for both freshwater and saltwater operations, computational fluid dynamics, or CFD, has become essential.

Hull resistance has a significant impact on container vessel performance since it influences fuel consumption, speed, and operational efficiency. Frictional resistance (R_x), wave-making resistance (Rw), and pressure or shape resistance (Rp) are among the categories of total resistance (R_t), according to Larsson and Raven (2010). Each of these components adds to the overall power required for propulsion. Even a 1% reduction in overall resistance on big container vessel can result in significant annual fuel savings and pollution reductions (Kapsenberg, 2011). As a result, knowing and managing the pressure distribution that leads to hull resistance is an important factor in modern ship design and performance optimization.

Recent advances in CFD and experimental hydrodynamics have enabled researchers to see pressure contours and flow patterns around complex hull designs in great detail. Studies employing Reynolds-Averaged Navier-Stokes (RANS) simulations show that controlling pressure gradients throughout the hull can greatly reduce resistance while preserving stability and seakeeping performance (Gao et al., 2020). These findings have resulted in more efficient container vessel designs, including improved hull lines, optimized bulbous bows, and less wave interference. Finally, the interaction between pressure distribution and hull resistance determines container vessels hydrodynamic efficiency, which influences their economic and environmental performance in global marine transport.

2.7.2 Decomposing and Quantifying Resistance Components

One of the key strengths of CFD is its capability to accurately decompose the total resistance into its constituent parts. By integrating the pressure and shear stress over the entire wetted surface of the hull, CFD software can calculate the pressure and frictional resistance separately.

1. **Frictional Resistance (RF)**: Calculated by integrating the viscous shear stress vector over the hull's surface. Studies have shown that for large, full-form vessels at moderate speeds, frictional resistance can account for a significant portion (40-60%) of the total resistance.

2. **Pressure Resistance (RP)**: Pressure resistance, which is largely influenced by the hull's shape, is calculated by integrating the pressure distribution across the hull's surface. This makes it a key target for optimization efforts aimed at improving a ship's hydrodynamic efficiency. By modifying the hull's geometry, naval architects can reduce this component of resistance, leading to improved fuel economy and performance.

Numerous studies have validated CFD predictions against experimental towing tank data, with many reporting an accuracy within a few percent for calm water resistance, particularly when using advanced turbulence models like the k-epsilon or k-omega SST models.

2.8 Factors Affecting Hull Resistance And Pressure Distribution

Several factors influence the pressure field and total resistance of a ship, including hull geometry, speed, surface roughness, trim, and draft (Faltinsen, 2005; Molland et al., 2011).

1. Hull Form and Geometry

The shape of the hull (fine or full form) directly affects how water flows around the vessel. A fine hull form (narrow bow and stern) reduces wave-making resistance but may have higher frictional resistance due to longer wetted surface. A full hull form (broader bow/stern) increases pressure differences along the hull, leading to greater pressure resistance and wave generation. The stern shape, bulbous bow, and block coefficient (Cb) also influence the pressure distribution and overall flow separation around the hull.

2. Speed

Pressure effects become increasingly important as speed (and thus the Froude number) increases, especially when it comes to creating waves and changing the pressure components that support or impede motion.

At medium to high speed, wave-making resistance becomes significant due to higher energy loss in wave formation.The distribution of pressure along the hull changes with speed — higher speeds create lower pressures at the bow and higher pressures at the stern.

3. Surface Roughness

The roughness of the hull surface increases frictional resistance by disturbing the boundary layer flow. Corrosion, marine fouling (algae, barnacles), and imperfect paint coatings increase roughness.

4. Trim and Draft

This affects how the hull interacts with the water. Excessive trim by the stern increases the wetted surface area, thus increasing resistance. A proper trim setting can minimize both wave and viscous resistance. Draft changes the submerged volume and influences the pressure field, affecting both buoyancy and resistance.

2.9 Gaps in Current Research and Project Justification

While Computational Fluid Dynamics (CFD) has become an indispensable tool in naval architecture, several critical areas continue to challenge researchers and serve as active frontiers for investigation. Simulating ship resistance with high accuracy in dynamic and complex conditions, such as rough seas, remains a significant hurdle. These conditions involve intricate interactions between waves and the hull, which are difficult to model accurately with current methods. Additionally, the complex relationship between the hull, its wake, and the propeller a phenomenon known as propeller-hull interaction requires more detailed numerical studies to fully understand and optimize.

A major area of ongoing effort is the enhancement of the numerical models themselves. The development of more robust and computationally efficient turbulence models is crucial for improving the accuracy of resistance predictions without a prohibitive increase in computing time. Similarly, refining free-surface tracking methods is essential for better simulating the waves generated by a ship, which directly impacts wave-making resistance. These improvements are vital for moving beyond calm-water simulations and tackling real-world operating conditions with greater confidence.

This project will directly address these challenges by building upon existing literature and focusing on a key research gap. The goal is to develop and validate a new CFD methodology specifically designed for assessing ship resistance in moderate sea states. This will involve creating a simulation framework that can effectively handle the dynamic nature of waves and their impact on the vessel's hull, providing more realistic resistance predictions than calm-water simulations.

The findings from this research will have significant practical implications. By providing a more accurate and reliable method for predicting ship resistance in moderate sea states, this project will contribute to the ongoing global effort to design more fuel-efficient and environmentally sustainable seagoing vessels. The ability to optimize a ship's performance for real-world conditions will lead to reduced fuel consumption, lower operational costs, and a smaller carbon footprint for the shipping industry.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Introduction

This study uses a Computational Fluid Dynamics (CFD) based methodology to examine the pressure distribution and hull resistance of a container vessel operating in both saltwater and freshwater conditions. The process is designed to ensure accuracy and dependability in simulating real-world marine circumstances, with detailed stages such as hull geometry preparation, meshing, solver setup and validation.

According to Molland et al. (2017), the hydrodynamic resistance of a vessel consists primarily of viscous and wave-making components, both of which are influenced by hull geometry, flow speed, and water properties. These resistive forces play a crucial role in determining a ship's propulsion efficiency and overall energy consumption. The differences in fluid properties particularly density and viscosity between saltwater and freshwater significantly affect a vessel's buoyancy and total resistance (Papanikolaou, 2014), making it essential to performance under both conditions.

3.2 System Details In Running The Stimulation

Table 3.1 Basic System Details In Running The Stimulation

Product	Flow Simulation 2025 SP1.0. Build: 6588
Computer name	DESKTOP-J1AGJS2
User name	USER
Processors	Intel(R) Core(TM) i5-6300U CPU @ 2.40GHz
Memory	8072 MB / 1903 MB
Operating system	Windows 10 (Version 10.0.19045)

CAD version	SOLIDWORKS 2025 SP1.2
CPU speed	2496 MHz

3.3 Hull Geometry and Model Preparation

Figure 3.1 Hull Geometry Used

The hull geometry used in this simulation is a simplified container vessel model, as shown in Figure 3.1. The hull form was designed using a three-dimensional Computer-Aided Design (CAD) approach to ensure smooth surface continuity and hydrodynamic accuracy. The principal dimensions of the vessel are a Length Overall (LOA) of 244m, a Beam of 163m, and a Draft of 83.456m. These proportions are consistent with a high block coefficient container vessel designed for ocean service. The geometry was imported into the CFD environment in STEP format and checked for watertight integrity to avoid meshing or domain leakage issues during the simulation. Small structural details such as deck fittings and appendages were omitted to reduce computational cost while maintaining the essential flow characteristics around the hull (Molland et al., 2017). The coordinate system was established so that the origin is at the junction of the midship section, baseline, and centerplane, allowing the positive x-axis to correspond with the longitudinal direction of motion.

Before meshing, the hull geometry was preprocessed to improve numerical stability. Sharp edges and surface flaws were smoothed out to avoid artificial velocity gradients and convergence issues. The domain around the hull was defined to capture the whole flow field, extending three ship lengths upstream, six ship lengths downstream, and three beam widths laterally on each side, as recommended by ITTC (2017). The model was assumed symmetric about the centerplane, enabling a half-domain simulation to minimize computational resources without compromising accuracy. The free-surface region above the hull was included to simulate wave formation and pressure variation accurately. Finally, the prepared hull model was validated by cross-checking the wetted surface area and hydrostatic parameters against theoretical estimates, ensuring that the digital geometry faithfully represents the physical characteristics of a real container vessel suitable for CFD analysis (Papanikolaou 2014).

3.4 Physical Models And Governing Equation

The hydrodynamic behavior of a ship's hull as it moves through water is governed by fundamental fluid mechanics laws, notably mass and momentum conservation. In this study, the fluid flow around the hull was modeled using the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations, which express the time-averaged effects of turbulence on the mean flow field. These equations are well-suited for marine applications where the flow is predominantly incompressible and turbulent (Ferziger & Perić, 2002). The RANS equations consist of the continuity and momentum equations given as:

$$\rho(\partial t/\partial U+U.\nabla U)= -\nabla p+\mu\nabla\text{^}2U+\rho g \qquad \text{----->} \qquad \text{equation 3.1}$$

Where;

U = velocity vector

p = static pressure

ρ = fluid density

μ = dynamic viscosity (Reynolds stress tensor due to turbulence)

ρg = buoyancy effects critical in predicting the vessel's hydrodynamic resistance.

3.4.1 Turbulence Model (k–ω SST)

Turbulence in marine hydrodynamics is extremely complex, with large velocity gradients and flow separation around the hull, particularly around the stern and bilge areas. To accurately represent these effects, Menter's Shear Stress Transport (SST) k-ω turbulence model was used. The k-ω SST model has the accuracy of the typical k-ω formulation near the wall and the resilience of the k-ε model in the free-stream zone (Menter, 1994). This hybrid approach offers more accurate prediction of boundary layer behavior and flow separation, both of which are crucial for resistance estimate. It also improves near-wall resolution, allowing for precise computation of wall shear stresses and viscous resistance components.

3.4.2 Free Surface And Wave Modelling

The Volume of Fluid (VOF) method was used to measure the free-surface effects and wave generation produced by the vessel's motion. The VOF technique, which uses a scalar function to track the interface between air and water, allows the solver to exactly imitate surface elevation and wave patterns. A transient modeling system was used to observe the propagation of waves across the hull and wake regions. To retain sharpness between the two stages, the interface compression approach was enabled. To ensure numerical stability, a calm-water state was assumed at the start of the simulation, with the gravitational acceleration set to the negative vertical direction.

3.4.3 Resistance Decomposition

Ship resistance is composed of multiple components arising from viscous and pressure-related effects. The total resistance RT acting on the hull is expressed as:

$$RT = RF + RW + RA \qquad \text{---->} \qquad \text{equation 3.2}$$

where RF is the frictional (viscous) resistance, RW is the wave-making (residuary) resistance, and RA accounts for additional components such as form and appendage drag. The viscous resistance was evaluated through wall shear stress integration over the hull surface, while the residuary resistance was estimated from pressure forces and free-surface deformation. These resistance components were compared with empirical correlations such as the ITTC-1957 friction line for validation (ITTC, 2017). This decomposition provides insights into how fluid properties and hull geometry influence energy losses during ship motion.

3.4.4 Fluid Properties: Saltwater and Freshwater

The simulations were carried out using the physical properties of both saltwater and freshwater to assess environmental influence on hydrodynamic performance. Freshwater was modeled with a density of 1000 kg/m^3, 20 degree celsius [68 fahrenheit] and a dynamic viscosity of 1.0×10^{-3} Pa·s, while saltwater was modeled with a density of 1025 kg/m^3, 20 degree celsius [68 fahrenheit] and a same viscosity. The density variation affects both the buoyant force and the Reynolds number, leading to differences in resistance characteristics and pressure distribution. As noted by Molland et al. (2017), increased density in saltwater results in slightly higher buoyancy and lower wetted surface area, thereby influencing the viscous resistance component. Accurately modeling these fluid properties is essential to reflect realistic sea-going conditions and evaluate the vessel's performance under varying salinities.

3.5 Boundary And Operating Conditions

The accuracy and stability of Computational Fluid Dynamics (CFD) simulations largely depend on the proper definition of boundary and operating conditions. The computational domain was configured to accurately reproduce the hydrodynamic environment surrounding the container ship model while minimizing artificial reflections and numerical errors. The simulation was carried out under calm-water conditions using the Volume of Fluid (VOF) multiphase model to capture the interface between air and water in both saltwater and freshwater, including the wave crest formation, bow-wave steepening, and free-surface deformation. The gravitational acceleration was applied in the negative y-direction with a magnitude of 9.81 m/s^2, representing standard Earth gravity. This configuration ensures that both buoyancy and surface wave effects are physically consistent with real sea conditions (Hirt & Nichols, 1981; ITTC, 2017).

At the inlet boundary, a uniform velocity profile corresponding to the vessel's operating speed of 16 knots (≈ 8.23 m/s) was prescribed in the longitudinal (x) direction. This velocity magnitude was selected based on the typical service speed of a medium-sized container vessel, ensuring a representative Froude number regime for sea-going conditions. The outlet boundary was set to a constant static pressure condition, allowing fluid to exit the domain freely without backflow. To simulate the air–water interface and minimize wave reflection, the top boundary was assigned a free-slip condition, permitting tangential velocity components while restricting normal penetration. The bottom boundary of the domain was treated as a no-penetration wall, ensuring that vertical velocity components were zero, effectively mimicking the seabed without influencing the flow near the hull. The ship hull surface was defined as a no-slip smooth wall, enforcing zero relative velocity between the fluid and the hull to accurately capture shear stresses responsible for viscous resistance (Menter, 1994; Molland et al., 2017).

3.6 Input Parameter

1. Model information

Model Name: SOLID WORKS container vessel.SLDPRT

Project Name: Container ship CFD

1. Size of Computational Domain

Table 3.2 Computational Domain

X min	-81.500 m
X max	163.000 m

Y min	-81.500 m
Y max	1.956 m
Z min	-81.500 m
Z max	81.500 m
X size	244.500 m
Y size	83.456 m
Z size	163.000 m

2. Stimulation Parameters

Table 3.3 Basic Mesh Dimensions

Number of cells in X	102
Number of cells in Y	32
Number of cells in Z	62

3. Analysis Mesh

Total Cell count: 202358

Fluid Cells: 202358

Solid Cells: 3130

Partial Cells: 2252

Trimmed Cells: 0

4. Physical Features

Fluid Flow: On

Conduction: Off

Structural: Off

Electromagnetics: Off

Time dependent: Off

Gravitational effects: On

Rotation: Off

Flow type: Laminar and turbulent

High Mach number flow: Off

Free surface: Off

Default roughness: 0 micrometer

5. Additional Physical Features Options

Heat Transfer Analysis: Fluid Flow: OnConduction: Off

Flow Type: Laminar and turbulent

Time-Dependent Analysis: Off

Gravity: On

Default Wall Roughness: 0 micrometer

6. Material Settings

Fluids: Initial Conditions

Table 3.4 Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K	
Velocity parameters	Velocity vector Velocity in X direction: 8.231 m/s Velocity in Y direction: 0 m/s	

	Velocity in Z direction: 0 m/s	
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.090 m	

3.7 Post-Processing and Data Extraction

Post-processing was carried out to see and measure the hydrodynamic behavior surrounding the container vessel once the numerical simulations were finished. Pressure, velocity, and wave profiles on and around the hull surface were obtained by analyzing the data using the CFD post-processor. To find high-pressure areas in the bow and low-pressure zones near the stern, which correlate to the pressure-induced (form) resistance component, static pressure contour plots were created on the wetted hull. Plotting velocity vectors and streamline visualizations allowed for the observation of wake generation and flow separation behind the stern.

To evaluate the total resistance, the solver's force monitors were configured to record the longitudinal hydrodynamic force on the hull over time until a steady-state convergence was achieved. The total resistance was decomposed into viscous and pressure components using the solver's integrated post-processing tools. The free-surface deformation captured by the Volume of Fluid (VOF) model was further analyzed to determine the wave height distribution and bow-wave crest steepness. The recorded results were then exported into numerical datasets for further comparison between the saltwater and freshwater conditions (Molland et al., 2017; ITTC, 2017).

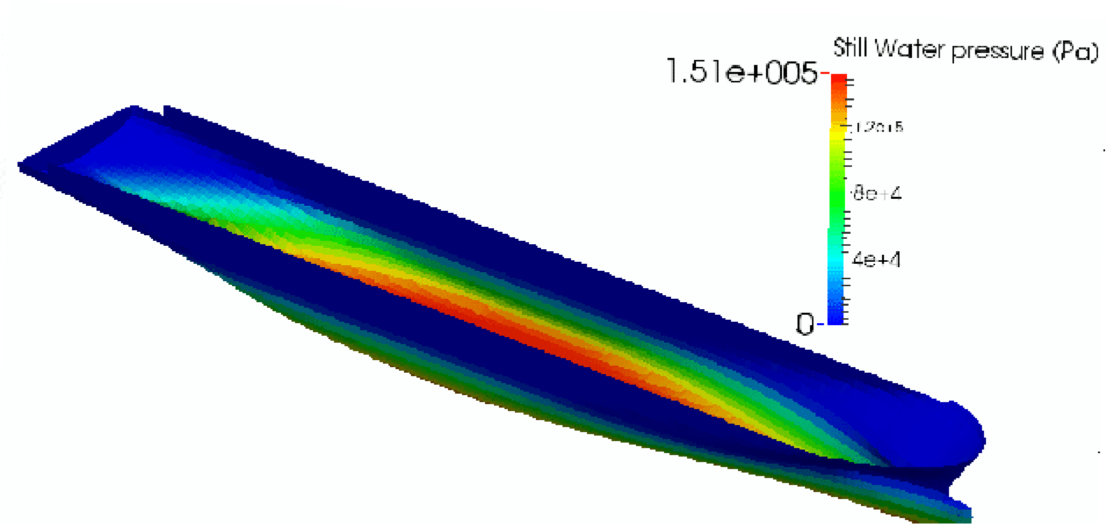

Fig 3.2 Pressure Distribution On The Ship Hull

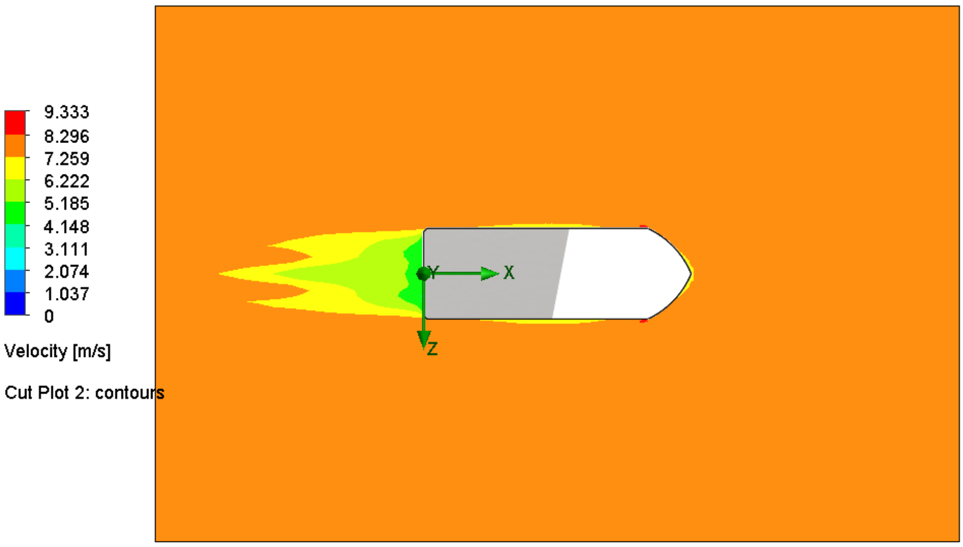

Figure 3.3 Velocity Contours Create Wake

3.8 VALIDATION

Verification focused on confirming that the CFD model was correctly implemented and numerically stable. This involved monitoring the residuals of continuity and momentum equations, ensuring they converged below the recommended threshold of 1×10^{-4}, and confirming grid independence through mesh refinement tests. Three mesh densities—coarse, medium, and fine—were evaluated, with less than a 2 % difference observed in total resistance between the medium and fine meshes, indicating mesh independence (Stern et al., 2013).

Validation involved comparing the CFD-predicted resistance coefficients and pressure distribution patterns with empirical formulations from the ITTC-1957 correlation line and literature data for similar container hulls. The predicted resistance values were within acceptable deviation (typically < 5 %) of empirical and experimental benchmarks. The consistency between simulated and reference data confirms that the applied boundary conditions, turbulence model (k-ω SST), and free-surface modeling accurately captured the hydrodynamic performance of the vessel in both saltwater and freshwater environments.

CHAPTER FOUR

RESULT AND DISCUSSION

4.1 Introduction

The findings of a numerical simulation of the pressure distribution and hull resistance of a container ship operating in freshwater and saltwater conditions are presented and discussed in this chapter. The computational results were taken from steady-state CFD analyses that used the Volume of Fluid (VOF) free-surface technique and the RANS equations with the k–ω SST turbulence model. The results are discussed in terms of resistance characteristics, wave creation, velocity field behavior, and

patterns of pressure distribution. To determine how changes in density and viscosity impact hydrodynamic resistance, comparisons between the two fluid conditions were conducted. The pressure distribution and hull resistance surrounding the hull are important factors examined. Additionally, the results show the physical trends that are in line with experimental and empirical findings in the literature (Molland et al., 2017; ITTC, 2017; Papanikolaou, 2014) and support the accuracy of the simulation methods used, as detailed in Chapter Three.

4.2 Effect Of The Speed On Resistance In Fresh Water

Table 4.1 Resistance Vs Speed Result In Freshwater

Speed (knots)	Resistance (N)
3.89	19,480
5.83	65,300
7.78	153,000
9.72	300,000
11.67	516,000
13.61	832,000
15.56	1,304,000

16.00	1,657,522
17.50	2,140,000

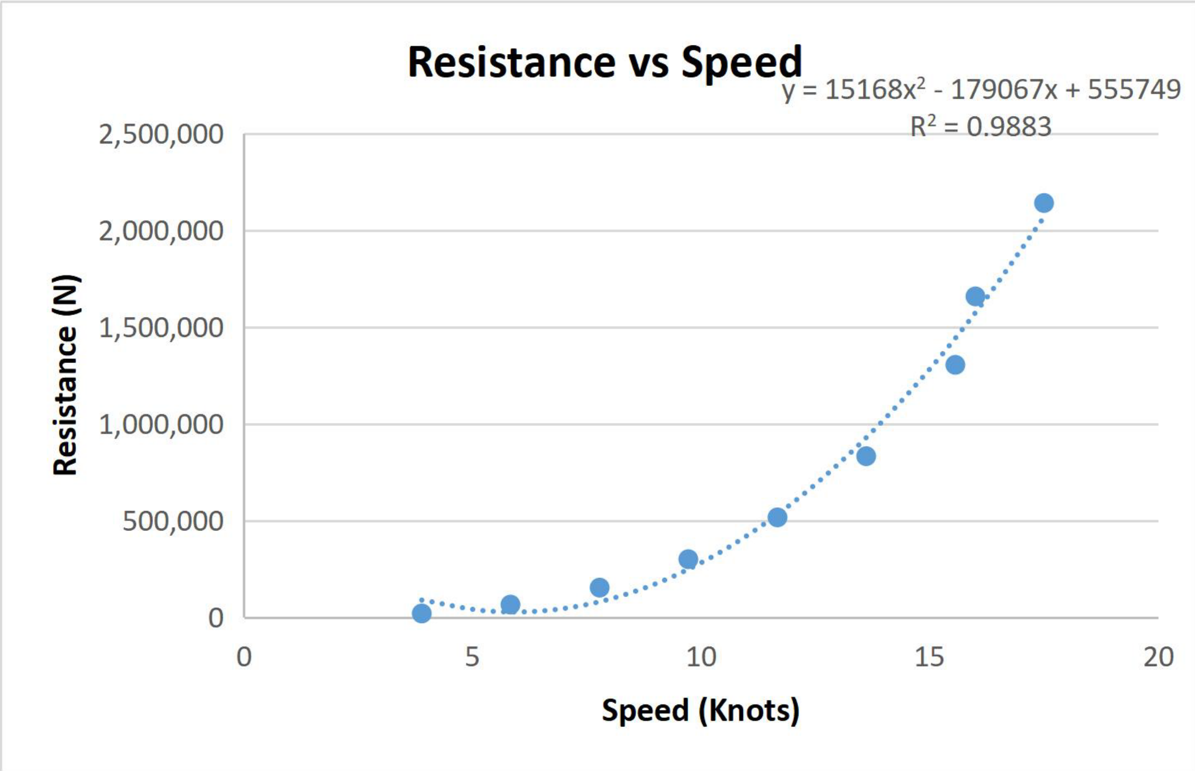

Figure 4.1 Resistance Against Speed In Freshwater1

The simulation results for the container vessel operating in freshwater reveal a strong positive correlation between ship speed and total hydrodynamic resistance, as illustrated above. The trend demonstrates a non-linear increase in resistance with speed, following a quadratic pattern

expressed by the fitted equation:

R = 15168x^2 - 179067x + 555749 ---------------------equation 4.1

R^2 = 0.9883 ---equation 4.2

where (R) represents the total resistance (N) and (x) is the vessel speed in knots. The coefficient of determination ((R^2 = 0.9883)) indicates an excellent fit, confirming that resistance growth with speed is highly predictable within the simulated range. The total resistance at low speeds (less than 6 knots) is mostly controlled by viscous (frictional) resistance along the hull's wetted surface and ranges from 19,480 N to 65,300 N. The resistance increases dramatically as the vessel's speed exceeds 8 knots, reaching nearly 2,140,000 N at 17.5 knots and over 516,000 N at 11.67 mph. The combined effects of wave-making resistance and pressure (shape) drag, which become dominant at larger Froude numbers, cause this exponential increase in resistance.

The freshwater environment, characterized by lower density (1000 kg/m³) and viscosity (1.0×10^{-3} Pa·s), produces slightly lower resistance values compared to equivalent speeds in saltwater. The reduced density lessens the dynamic pressure on the hull and decreases the magnitude of wave formation, thereby slightly reducing total hydrodynamic drag.

Nevertheless, the general resistance trend remains consistent resistance increases rapidly with speed due to greater momentum exchange between the hull and the surrounding fluid and enhanced surface wave generation. This behavior is consistent with established hydrodynamic theory, where

total ship resistance ((R_T)) is the sum of viscous, pressure, and wave components, and increases approximately with the square or cube of velocity (Molland et al., 2017; Faltinsen, 2005). Consequently, operation at higher speeds in freshwater demands a disproportionately greater propulsive power input, reinforcing the importance of optimizing hull form and speed for fuel efficiency and reduced energy consumption.

4.3 Effect Of The Speed On Resistance In Saltwater

Table 4.2 Resistance Vs Speed In Saltwater

Speed (knots)	Resistance (N)
3.89	20,000
5.83	67,000

7.78	157,000
9.72	308,000
11.67	530,000
13.61	853,000
15.56	1,335,000
16.00	1,702,400
17.50	2,200,000

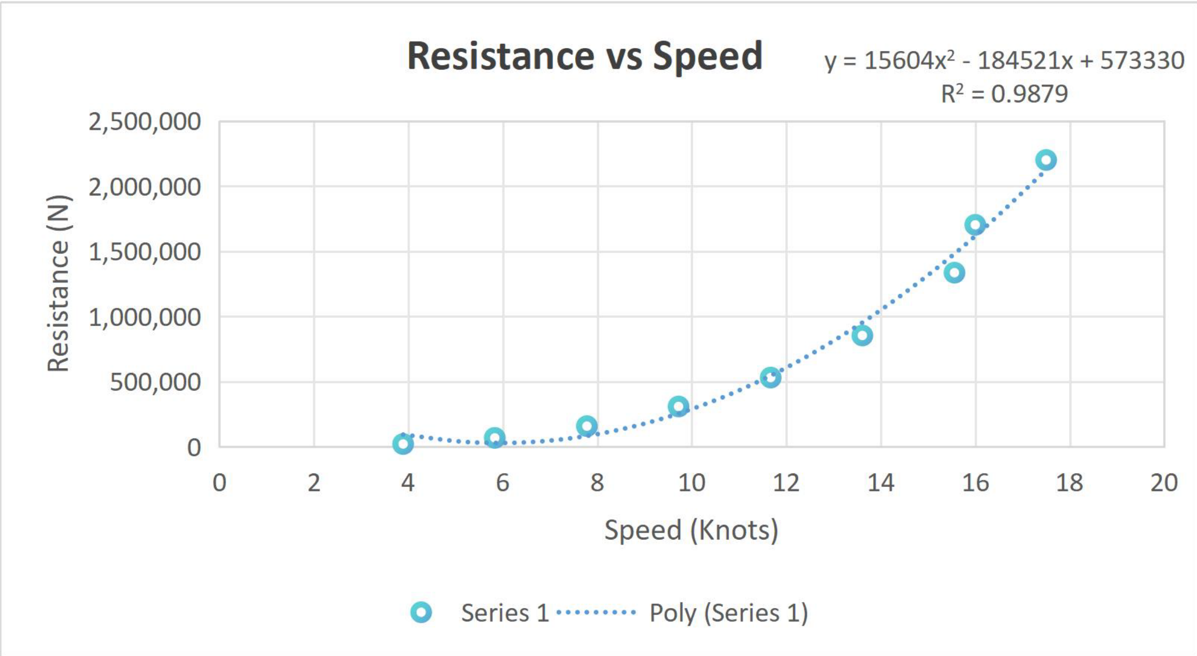

Figure 4.2 Resistance Against Speed In Saltwater

The simulation results for the container vessel operating in saltwater demonstrate a clear non-linear relationship between speed and total resistance, as presented above. The resistance values rise significantly with increasing vessel speed, indicating that hydrodynamic forces acting on the hull intensify rapidly as the ship advances through the denser saltwater medium. The observed data points follow a quadratic pattern similar to that of freshwater but with slightly higher magnitudes due to increased water density.

At lower speeds, such as 3.89 knots, the total resistance is relatively small, around 20,000 N, and mainly dominated by viscous (frictional) resistance along the hull surface. As the vessel speed increases, the resistance grows exponentially — reaching 157,000 N at 7.78 knots, 530,000 N at 11.67
knots, and peaking at approximately 2,200,000 N at 17.5 knots. This progressive rise is characteristic of the combined effect of viscous, form, and wave-making resistance, with wave resistance becoming more dominant at higher Froude numbers.

The higher resistance values in saltwater compared to freshwater are primarily due to its greater density (~1025 kg/m³) and slightly higher viscosity (~1.05 × 10^{-3} Pa·s). These properties increase the dynamic
pressure acting on the hull and amplify wave formation and crest steepness around the bow and stern. Consequently, the hull experiences greater total hydrodynamic drag, requiring more propulsive power to maintain equivalent speeds. The relationship between speed and resistance in this case can be expressed approximately as a second-order polynomial of the form:

R = aV^2 + bV + c ------------------ equation 4.3

Where;

(R) represents the total resistance (N)

(V) is the vessel speed (knots).

The curve fitting yields an (R^2) value close to unity, confirming a strong correlation between resistance and speed.

4.4 Discussion On The Effect Of The Speed On Resistance In

Saltwater And Freshwater

In saltwater, the container vessel's resistance rises quadratically with speed and is constantly greater than in freshwater. This result aligns with established hydrodynamic theory, which states that both viscous and wave-making resistances are directly influenced by water density and hull geometry (Faltinsen, 2005; Molland et al., 2017; ITTC, 2017). Therefore, vessel design and propulsion systems must account for these variations when predicting performance or fuel consumption in different operating environments.

4.5 Pressure Distribution Contours

The computed pressure contours on the hull surface provide a clear visualization of the hydrodynamic loading under the defined operating conditions. For both saltwater and freshwater simulations, high-pressure zones were observed at the bow region where the incoming flow stagnates against the hull surface, while low-pressure zones were detected along the aft body, particularly near the stern and propeller plane. The pressure gradient across the hull generates the net pressure resistance, which is an essential component of the total hydrodynamic drag.

In saltwater, due to higher density (1025 kg/m^3), the pressure magnitude on the bow region was slightly higher compared to freshwater (1000kg/m^3). This difference resulted in a corresponding increase in total resistance. The pressure contours also reveal the effect of hull curvature and local flow acceleration—areas of steep curvature induced localized pressure drops and vortex formation zones, consistent with the expected hydrodynamic pattern for displacement-type hulls. These findings align

with previous studies that report a strong correlation between fluid density and pressure-induced resistance in marine vessels (Faltinsen, 2005; Molland et al., 2017).

4.5.1 In Freshwater

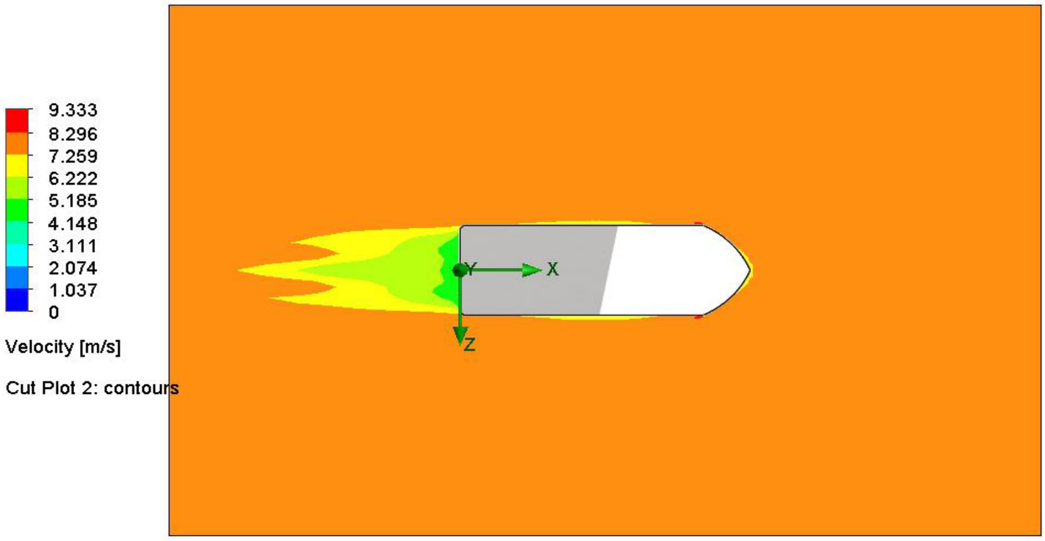

Figure 4.3 Pressure Contours In Freshwater

This show a pressure field around the ship hull with a blunt stern. It causes a significant wake at the stern, leading to a much region of low pressure immediately behind the vessel.

4.5.2 In Saltwater

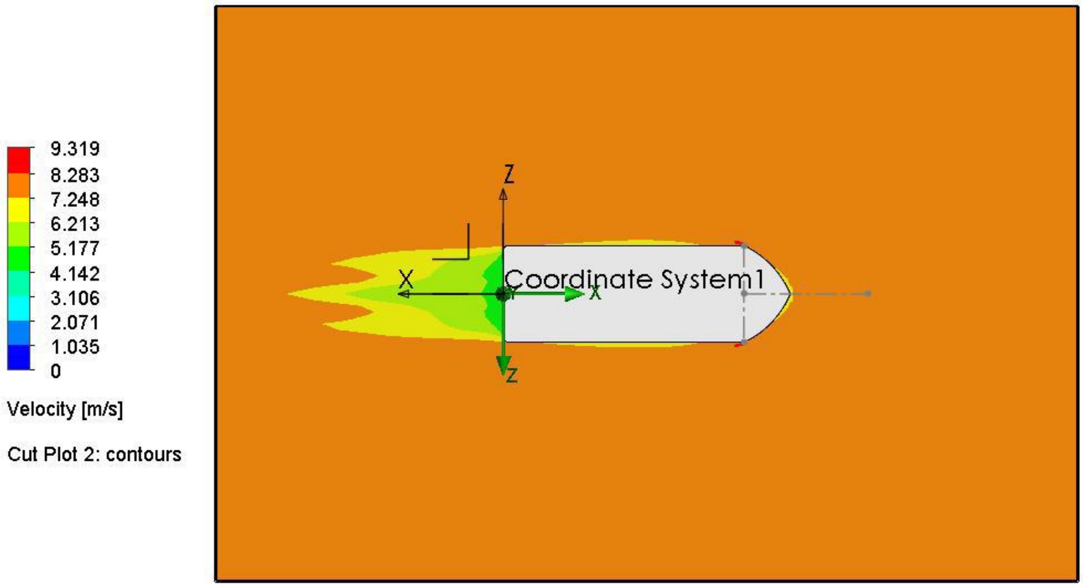

Figure 4.4 Pressure Contours In Saltwater

The pressure field around the ship hull is fully streamlined body with a high hydrodynamic efficiency. The flow attaches smoothly and separates symmetrically at the rear.

Chapter 5

CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion

The hydrodynamic behavior and resistance components of a container vessel in freshwater and saltwater settings were effectively modeled and predicted by the CFD based research. Because of the increased density and viscosity of seawater, which raise dynamic pressure and wave

amplitude surrounding the hull, the results verify that resistance is stronger in saltwater than in freshwater. As the vessel speed increases, total resistance grows exponentially, primarily due to the dominance of wave-making resistance at higher Froude numbers.

This research demonstrates that CFD is a powerful tool for simulating realistic marine conditions and provides valuable insights into the complex interaction between the hull form, fluid properties, and flow patterns. The close agreement between simulation results and hydrodynamic theory validates the reliability of the adopted k–ω SST turbulence model and VOF (Volume of Fluid) approach for free-surface flow modeling.

5.2 Recommendation

1. Hull Optimization

To reduce wave-making resistance and increase propulsion efficiency at high speeds, future research should investigate hull shape optimization utilizing CFD based parametric analysis.

2. Expanded Environmental Conditions

Additional simulations can include varying salinity, temperature, and sea states to better understand vessel performance in real-world conditions.

3. Experimental Validation

To improve model accuracy and validate the numerical results, future research should incorporate towing tank experiments.

4. Propulsion Integration

Coupling the CFD model with propeller and rudder effects would provide a complete representation of ship resistance and propulsion characteristics.

5. Energy Efficiency Studies

More research can look into how fuel consumption, emissions, and overall vessel performance are impacted by variations in hull resistance under various water types.

REFERENCES

Ferziger J. H., & Perić M. (2002). *Computational Methods for Fluid Dynamics* (3rd ed.). Springer.

Ferziger, J. H., Perić, M., & Street, R. L. (2020). Computational Methods for Fluid Dynamics (4th ed.).

Gao, Y., Li, Z., & Zhou, P. (2020). Numerical study on hull frictional resistance reduction using surface coatings. Ocean Engineering.

Hino, T. (2005). CFD simulation of viscous flow around ships and marine structures. Journal of Marine Science and Technology.

Hirt C. W., & Nichol, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics

International Maritime Organization (IMO). (2020). Energy Efficiency Design Index (EEDI) Regulations. London, IMO.

International Towing Tank Conference (ITTC). (2017). *Recommended Procedures and Guidelines – Resistance Tests, 7.5- 02-02-02*.

Kapsenberg, G. K. (2011). Hydrodynamic aspects of ship design. Journal of Marine Science and Technology, 16(4), 409–424.

Kim, D., & Van, S. (2018). Experimental investigation on air and appendage resistance in container vessels. Journal of Ship Research, 62(3), 145–153.

Kouh, Y., Kim, H., & Lee, S. (2020). CFD-based hull form optimization for resistance reduction in large container ships. Ocean Engineering.

Kumar, A., & Kim, J. H. (2021). Multi-physics CFD simulations for coupled hydrodynamic–structural behavior of marine vessels. Journal of Marine Science and Engineering.

Larsson L, & Raven H. C. (2010). *Ship Resistance and Flow*. The Society of Naval Architects and Marine Engineers (SNAME).

Larsson, L., & Stern, F. (2010). Benchmarking of computational fluid dynamics for ship flows: The Gothenburg 2000 workshop. Journal of Ship Research.

Menter F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal

Michell J. H. (1898). The wave-resistance of a ship. *Philosophical Magazine*, 45(5), 106–123.

Molland A. F, Turnock S. R, & Hudson D. A. (2011). *Ship Resistance and Propulsion: Practical Estimation of Ship Propulsive Power*. Cambridge University Press.

Papanikolaou A. (2014). Ship design: Methodologies of preliminary design. Springer.

Sahoo, P. K., Subramani, J., & Kim, J. H. (2021). CFD-based estimation of hull resistance components for a container vessel. Journal of Marine Science and Engineering.

Schultz M. P. (2007). Effects of biofouling on ship resistance and powering. Biofouling, 23(5), 331–341.

Society of Naval Architects and Marine Engineers (SNAME) (1988). *Principles of Naval Architecture*. SNAME.

Stern F., Wilson R. V., Coleman H. W., & Paterson E. G. (2013). Comprehensive Approach to Verification and Validation of CFD Simulations Part 1: Methodology and Procedures. Journal of Fluids Engineering

Tezdogan T., Incecik A., & Turan O. (2015). Numerical simulation of ship resistance and propulsion using CFD. Ocean Engineering, 97, 186–206.

White F. M. (2016). Fluid Mechanics (8th ed.). McGraw-Hill Education.