UNIVERSITI TEKNOLOGI MARA
CAWANGAN TERENGGANU

MEC299

RESISTANCE PREDICTION OF AMPHIBIOUS BOAT IN CALM WATER USING COMPUTATIONAL FLUID DYNAMIC APPROACH

NABILATUL ATHIRAH BINTI MISNAN

2020613788

SUPERVISOR:

MISS NUR AIN BINTI ABD RAHMAN

SEM MARCH AUGUST 2022

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

1.1 Background of Study

Humans' continual exploration of space has resulted in significant breakthroughs in marine and terrestrial technology. As one of the world's most populous countries, China has many rivers and lake. There are almost 1500 rivers in China with a drainage area. There are over 2800 lakes and over one hundred coastal bays in the area. The drainage area is home to the ethnic group. Water and land transportation is in high demand from people worldwide as society develops. Rather than being intended for land transport with the capacity to cross water, amphibious vehicles are designed for water transport with the ability to go on land. The difference is that the vehicles are meant to be high performing on water, with land movement provided as a bonus feature rather than as the primary role.

The amphibious platform can help individuals move more efficiently by improving the efficiency of their water or land conveyance and reducing trip time. Water covers approximately 75% of the Earth's surface. A vehicle that can move on both land and water has the potential to disrupt the current transportation model. Land transportation is quite widespread, but waterways are naturally available but underutilised, and Amphibian vehicles have shown to be effective in this regard. One of the most crucial parts of a vehicle's performance is its design. When a vehicle travels at a high speed through a medium, the medium exerts resistance on the vehicle.

The force required to tow or move a ship in calm water at a steady pace is known as ship resistance. The resistance of a ship is proportional to its speed and velocity. One of the most significant aspects of a ship's design is its resistance. Computational Fluid Dynamics (CFD), a simple and time-saving application, can be used to determine the ship's resistance. The CFD analysis' correctness has been demonstrated. Engineering power generation industries are heavily depending on CFD to decrease development and retrofitting costs due to concerns about global warming and a rising worldwide population. CFD is a cost-effective alternative to experimental and analytical methodologies for modelling real fluid flows.

1.2 Problem Statement

Amphibious boats have planned hulls that are set up to create positive dynamic bottom pressures at high speeds. These positive pressures raise the hull, which lowers the buoyant part of hull support. Therefore, when planning, the wetted bottom area is much smaller than the static wetted area. Unfortunately, the addition of induced drag associated with the formation of dynamic lift leads to a total resistance weight ratio that is significantly higher than that of a displacement or semi-displacement vessel at their design speeds. High-speed planning hulls also provide hydrodynamic lift, which lowers wave-making resistance and might produce spray at the bow zone. Because it is closely related to the geometry, trim, and speed of the hull, the spray should also be considered and visualised in the early stages of ship design. Ship resistance will be broken down into several components, including resistance from friction, pressure resistance from viscosity, and resistance from waves.

1.3 Objectives

The main objective of this project is:

i. To analyse resistance of the amphibious boat in calm water using computational fluid dynamic approach.

ii. To determine frictional and pressure resistance of the amphibious boat.

1.4 Scope of Work

For this project, the experiment will focus on simulation by using any software such as computational fluid dynamic (CFD). The results of this experiment will be analysed for ship resistance and pressure distribution. The development of more user-friendly CFD-based software packages allows for visual reinforcement of fluid flow ideas. The analysis of systems involving fluid flow, heat transport, and related phenomena including chemical reactions using computer-based simulation is known as computational fluid dynamics, or CFD. Application of CFD is fast growing in ship hydrodynamics. Main advantage of CFD analysis is its flexibility and capacity to fulfil both Froude and Reynold similarities when towing tank approach is unable to fulfil Reynold similarity. This necessitates the search for numerical solutions to partial differential equations, and it appears, in retrospect, that the use of linear algebra for numerical method classification is erroneous.

1.5 Significance of Study

CFD simulation at full scale can be employed as a trustworthy and efficient method for resistance performance. The approach benefits from directly modelling the ship at full scale. Therefore, extrapolating from a model scale to a full scale is not necessary. This is crucial, especially for novel and unusual hull designs with unidentified form factors. As a result, there is a lot of ambiguity when determining hull resistance through towing tank testing. This problem is resolved by full-scale CFD simulations, which also have the advantage of assessing a wide range of hull forms and evaluating them from both an evolutionary and a novel perspective.