Incoming Wave Influence on Amphibious Vehicle Sailing Characteristics

Olarn Koonsombatkul [1*], Artit Ridluan [1]

[1] Research and Development, Defence Technology Institute (Public Organisation), Nonthaburi, Thailand
*Corresponding Author: Olarn.K@dti.or.th

ABSTRACT

For the design of amphibious system, understanding the hydrodynamics behavior and loads generated, while the vehicle drives through the water is important. Calculation using the traditional theory is difficult to accurately estimate the water resistance forces and load distribution. Therefore, this present work applied the computational fluid dynamic (CFD) to study hydrodynamics behavior and calculated water resistance force as well as a aerodynamic forces. The VOF model together with *wall-adapting local eddy* model was used to predict behavior and loads. From the numerical results, the separation zone behind the amphibious vehicle platform is observed and the water force distribution while the vehicle is driving through the water.

Keywords: Amphibious vehicle, wave making resistance, Volume of Fluid (VOF) model

1. Introduction

The flow field around amphibious vehicle is dominated by wave on the free surface between air and water. The flow field is turbulent that presented by separation and recirculation regimes. Hydrodynamics behavior of amphibious vehicle is therefore a complex fluid dynamics problem analyzed by experimental and numerical simulation. Recently, advances in computer technologies enable engineers to use computational fluid dynamics techniques to resolve efficiently the hydrodynamics problems.

Many researchers attempted to use CFD code to investigate the behavior of ship hull and load generated. The main problem encounter is that there are two different phases that should be considered, air and water, which is called free surface. Free surface flows occur whenever water comes in contact with air. A lot of studies have been needed to understand the physical flows. The incompressible free surface flow around the ship hull model has been studied using finite volume flow commercial code, FLUENT and CFX. The analysis considered two different models, standard k-ε and standard k-ω [1-3, 5-10].

There are difference between ship and amphibious vehicle such as shape, size, weight and wetted area that cause the flow around the hull changes. They affect hydrodynamics load generated. Nowadays, the hydrodynamics applied to amphibious vehicle is not yet presented. Therefore, we apply hydrodynamics principle of ship to study the behavior of vehicle and to predict the hydrodynamics load using CFD technique.

The objective of this study is to investigate the free surface flow around amphibious vehicle. The frictional resistance and the pressure resistance coefficient of the hull have been calculated numerically. The free surface flow generation mechanism and its interaction with the vehicle are described.

This paper was organized as follows. The governing equations and turbulence models are firstly briefed and follow by the description of the CFD simulation and configurations of