

Computational Investigation of Armored Vehicle Hydrodynamics

Olarn Koonsombatkul and Artit Ridluan

Operation Military Vehicle, Defence Technology Institute (Public Organization), Thailand

ABSTRACT

Armored vehicle is widely used in recent warfare since it can perform with variety of operations. One of those is amphibious. For the design of amphibious system, we need to understand the hydrodynamics behavior and loads generated while the vehicle move on the surface. Therefore, this present work applied the computational fluid dynamic (CFD) to study hydrodynamics behavior and calculate the hydrodynamics as well as aerodynamic forces. The VOF model together with the standard k- ϵ model was used to predict hydrodynamics behavior and loads. From the numerical results, the separation zone behind the armored vehicle platform is observed and there is the wave generated while the vehicle is moving through the water.

Keywords: Free surface Flow, Hydrodynamics Simulation, Volume of Fluid (VOF) Model

1. INTRODUCTION

The flow field around ship 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 ship 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 [1-7].

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 amphibious vehicle hydrodynamics to the free surface flow around armored vehicle model. The frictional resistance and the pressure resistance coefficient of the hull model 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 the amphibious vehicle. Both hydrodynamic coefficients and flow structures are investigated and presented in the results and discussion. Finally, the conclusion would be made.