View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by UMP Institutional Repository

Colloquium on Robotics, Unmanned Systems And Cybernetics 2014 (CRUSC 2014)
Nov. 20, 2014 at Universiti Malaysia Pahang, Pekan, Pahang, Malaysia

# CFD Analysis for Rigid Moving Body at the High Tidal Environment of Sea-bed

Md. Moktadir Alam[1], and Addie Irawan[2]

[1,2] Robotics and Unmanned Systems (RUS) Group, Faculty of Electrical and Electronics Engineering, Universiti Malaysia Pahang, Pekan, 26600, Malaysia

(E-mail: moktadir.alam@hotmail.com[1], addieirawan@ump.edu.my[2])

***Abstract*** - In this paper, the basic hydrodynamic theories have been used to find the hydrodynamic factors for the underwater moving rectangular body as considered the shape of underwater walking robot. The added-mass, wave drag and lift coefficients are determined using a frequency-domain, simple-source based boundary integral method. In this paper, the hydrodynamics added mass and drag forces will be determined theoretically using Buckingham π theorem. Through numerical calculation the Reynolds number is measured in order to understand the type of water flow over the structure. The relative velocity vectors, Reynolds number, drag and lift forces for each state of motion is obtained in both static water condition and in ocean current condition. Results are obtained for a range of wave frequencies and depths of the underwater robotic body submerged all for a fixed water depth of 50-100 m. With the wave exciting force and moment determined using the Navier-Stokes theory. The computational study is to determine body-shape effects on the incident and radiated wave forces and subsequently the motion response. This study and results further implemented to modern adaptive drag force model-based controller in horizontal flow disturbance control for underwater multilegged or wheeled robot.

***Keywords*** - CFD, Neiver-Strokes, Hydrodynamics, Reynolds Numbers.

## 1. Introduction

In the recent years, the applications of autonomous underwater vehicles (AUVs) have been expanding and now include fields such as marine research, technical investigations, deep-sea development and undersea projects. Depending on the underwater robotic body size and shape the AUV's can be divided into two categories and the shape is determined by the requirements of the robot's operation and task. For an instance, a streamlined shape AUV is able to reduce water resistance and is preferable if the vehicle must move at high speeds. Whereas, if underwater detection or operation tasks are the primary roles of an underwater robot, a non-streamlined shape is often used. Deep-sea research requires high water-pressure resistance, while monitoring and observation tasks require small, flexible and stable robots[1].

A dynamics model is necessary in the design of guidance, navigation and control (GNC) systems of an underwater robotic system. The need for performance improvements in GNC has motivated deeper investigation into hydrodynamics modeling and the identification of the robot dynamics. However, the identification of hydrodynamics parameters is very difficult and challenging as reported[2]. It is complicated to tune an exact and comprehensive dynamics model for an underwater walking robot due to the hydrodynamics parameters. In particular, the hovering and tight maneuvering motions of an underwater vehicle is difficult to characterize hydro-dynamically as the hydrodynamic drag and lift forces relates the interaction between the robotic object and the fluid surrounding which is very robust and changes based on the system dynamics as well as the environmental situations[3, 4]. Researchers have adapted various numerical approaches such as Lagrange method based Morison equation, DATCOM method, Roskam method, kinetic energy theorem based on fluid mechanics theory to model the appropriate hydrodynamic profile[5] for underwater robotics. Currently, there are many Computational Fluid Dynamics (CFD) software packages like ANSYS, FLUENT, OPENFLOW to study AUVs reaction to hydrodynamic forces and flow simulations[6, 7].

In this paper, the hydrodynamic drag and lift forces will be determined numerically using the continuity equation for unsteady motion of a viscous compressible fluid having variable physical properties and then verified with