A CFD (Computer Fluid Dynamics) Analysis Based Design Method For An Autonomous Underwater Vehicle Ducted Propeller

Tae-hwan Joung, Karl Sammut, and Fangpo He

School of Computer Science, Engineering and Mathematics, Faculty of Science & Engineering, Flinders University, Adelaide, South Australia, Australia

ABSTRACT

This paper presents a method for estimating the propulsion performance and efficiency of an AUV thruster based on CFD (Computer Fluid Dynamics) analysis. A discussion of the factors influencing the numerical simulation accuracy and convergence rate, namely the number, size, and type of mesh elements required to describe each part of the model, is presented. The axial symmetry characteristics of the propeller, nozzle and the encompassing MRF (Moving Reference Frame) region have been used to reduce complexity and hence computation time. The paper, then, describes a method for estimating the motor horsepower and RPM of an AUV thruster based on simulated resistance tests and propeller open water (POW) tests. The engine horsepower for an AUV equipped with a ducted propeller and RPM is evaluated from the speed-power curve. The effectiveness of the ducted propeller is studied by comparing the performance characteristics for test results obtained without the nozzle versus those obtained for a range of nozzle shapes. The best design of the nozzle shape is finally determined from speed-power curves of the given duct.

KEY WORDS: AUV (Autonomous Underwater Vehicle); CFD (Computational Fluid Dynamics); Nozzle propeller, Propeller Open Water test; Self propulsion test.

INTRODUCTION

An understanding of the fluid interaction between the hull and propeller is necessary to estimate propulsion performance and efficiency of an Autonomous Underwater Vehicle (AUV) thruster. Resistance and POW (Propeller Open Water) tests are essential preliminary steps that are required to determine hull and propeller efficiency. Self-propulsion testing can then be carried out to estimate the required power as well as obtain the relative rotative efficiency (η_h), the RPM, and the propulsion factors such as thrust deduction factor (t) and wake fraction factor (w). In order to increase efficiency the propeller thrust coefficient (C_{Th}) must be reduced. This can be facilitated by installing a duct around the propeller. The ducted propeller concept was first introduced by Kort (1934), and has been widely researched during the 1950s and 1960s (Manen and Oosterveld, 1966; Wessinger and Maass, 1968). Profound research about its shape and characteristics has been conducted based on the Netherlands Ship Model Basin (Oosterveld, 1973). The application of ducted propeller has been particularly adopted for use in low speed vessels such as tugboats and trawlers (Lewis, 1988).

The particular feature of the ducted propeller is that its efficiency is relatively low at high speed (over 15knots) due to the added drag of the nozzle. Therefore, the ducted propeller is prevalently used as a ROV or AUV thruster, not just to protect the propeller from contact with other objects but also because its power efficiency can be maximized for low speeds. Furthermore, efficiency of the ducted propeller at low speeds is highly appropriate for an AUV which has a low occurrence of cavitation and a need for high thrust power at low speeds.

In this paper, the CFD tool, ANSYS-CFX has been used to conduct resistance tests, POW (Propeller Open Water) tests, and self propulsion tests in order to predict the speed-power characteristics of an AUV. The required effective power (P_E) relative to the AUV velocity has been estimated by conducting resistance test on the AUV body. Subsequently, various CFD analysis methods were investigated to verify their reliability, prior to conducting the POW test to predict the efficiency of the propeller.

The POW test was initially carried out for the propeller without nozzle and then with a variety of nozzle shapes so that the effectiveness of the duct can be observed with the verified CFD analysis method. Finally, the speed-power predictions have been computed out by means of the ‘ITTC 1978 performance prediction method’ based on the resistance test and the POW test to find the best design of the nozzle shape. The values of Thrust coefficient (K_T), Torque coefficient (K_Q) and Open-water efficiency (η_O) obtained from the POW test have been used to estimate the propeller efficiency and predict the speed-power characteristic of the AUV. The propulsion factors for the speed-power predictions, such as relative rotative efficiency (η_h), thrust deduction factor (t) and wake fraction factor (w) are estimated first by approximation and then determined from the CFD based self-propulsion test.

RESISTANCE TEST

In order to estimate the drag force of the AUV hull, a resistance test has been carried out by means of the CFD code. The total drag force estimated by the CFD analysis has been verified by the ‘ITTC 1957