IJNAOE
©SNAK, 2014

Int. J. Nav. Archit. Ocean Eng. (2014) 6:269~281
http://dx.doi.org/10.2478/IJNAOE-2013-0178
pISSN: 2092-6782, eISSN: 2092-6790

Verification of CFD analysis methods for predicting the drag force and thrust power of an underwater disk robot

Tae-Hwan Joung[1], Hyeung-Sik Choi[2], Sang-Ki Jung[2], Karl Sammut[1] and Fangpo He[1]

[1]School of Computer Science, Engineering and Mathematics, Flinders University, Adelaide, Australia
[2]Division of Mechanical and Energy Systems Engineering, Korea Maritime University, Busan, Korea

ABSTRACT: *This paper examines the suitability of using the Computational Fluid Dynamics (CFD) tools, ANSYS-CFX, as an initial analysis tool for predicting the drag and propulsion performance (thrust and torque) of a concept underwater vehicle design. In order to select an appropriate thruster that will achieve the required speed of the Underwater Disk Robot (UDR), the ANSYS-CFX tools were used to predict the drag force of the UDR. Vertical Planar Motion Mechanism (VPMM) test simulations (i.e. pure heaving and pure pitching motion) by CFD motion analysis were carried out with the CFD software. The CFD results reveal the distribution of hydrodynamic values (velocity, pressure, etc.) of the UDR for these motion studies. Finally, CFD bollard pull test simulations were performed and compared with the experimental bollard pull test results conducted in a model basin. The experimental results confirm the suitability of using the ANSYS-CFX tools for predicting the behavior of concept vehicles early on in their design process.*

KEY WORDS: Underwater disk robot (UDR); Computational fluid dynamics (CFD); Drag force; Planar motion mechanism (PMM) test simulation; Bollard pull test; Thrust; Torque.

INTRODUCTION

A large variety of underwater robots have been developed by many research institutes over the last two decades. Some of the more successful of these robots are now being employed for scientific, commercial and military purposes. Research into developing more efficient underwater robots is being conducted with a view to improving mission duration (Yuh, 2000).

A new type of the underwater robot called an UDR has been developed and its design evaluated by CFD analysis such as a resistance test, propulsion test (bollard pull test) and Planar Motion Mechanism (PMM) test simulation. The body of the UDR is designed as a disk shaped vehicle in order to minimize the effect of external disturbances such as currents and waves. The UDR is composed of hull and frame structure, three vertical thrusters, three horizontal thrusters, a control system and sensors. The thrusters are mounted axi-symmetrically at an angle of 120 degrees to enable the UDR to navigate along any direction by vector summation with the propulsion controller.

Resistance (drag) testing, Propeller Open Water (POW) testing and PMM testing are essential steps that are required to predict the drag force, propulsion power and motion performance of an underwater vehicle. Conventionally, the tests for the prediction of drag, propulsion performance and motion of an underwater vehicle are carried out in a large model basin equipped with a towing carriage, and dynamometer, making the test process expensive. The development of commercial codes for CFD

Corresponding author: *Tae-Hwan Joung*, e-mail: *cpjoung@naver.com*

analysis now make it possible to predict drag and propulsion performance of a ship or submersible vehicle such as an underwater robot without using a physical model test basin (Joung et al., 2012).

There are many efforts at predicting drag forces of underwater vehicles by CFD analysis and validating CFD simulation methods in design processes (Yu and Su, 2010, Bellingham et al., 2010). CFD simulations are also compared with in-service data for the self-propelled performance of an Autonomous Underwater Vehicle (AUV) (Phillips et al., 2008). However, carrying out experimental tests for ducted propellers of underwater vehicles, or predicting performance of an UDR, is rarely reported.

In the work reported in this article, CFD analysis was first used to conduct the resistance test necessary to predict the total drag force for selecting an appropriate thruster that will achieve the required speed of the UDR. Pure heaving motion and pure pitching motion studies were then carried out to emulate the VPMM test by CFD motion analysis. The hydrodynamic forces on the UDR body and the distributions of the hydrodynamic values (velocity, pressure etc.) around the UDR body were obtained while the UDR was performing these motions.

Finally, the custom designed ducted propulsion system that is employed in the UDR was also analysed using CFD tools. The results of the CFD analysis for the ducted thruster were compared for validation purposes with the experimental test results that were obtained using a specially designed thrust measurement system. In order to further verify the validity of the CFD modeling process, a CFD model of a commercial thruster was developed and analysed and the estimated thrust performance characteristics compared against the corresponding physical test data as supplied by the thruster manufacturer.

DRAG ESTIMATION OF THE UNDERWATER DISK ROBOT

Concept design (initial design)

The UDR, shown in Fig. 1, was designed as a streamlined disk shape to reduce the drag force on the body in the horizontal direction. The bare hull of the UDR measures 1.9*m* in diameter and 0.45*m* in height.

The design speed (NCR; Normal Continuous Rating) for the initial design of the UDR is 3.5*knots* (1.8004*m/s*) and the maximum speed (MCR; Maximum Continuous Rating) is 5*knot* (2.572*m/s*). The configuration of the thrusters is shown in Fig. 2. The three horizontal thrusters, which were custom designed, and manufactured by the Korea Maritime University, are mounted in the x, y plane at intervals of 120° to each other about the z-axis, so that the UDR can move laterally by their resultant force. The three horizontal thrusters can also rotate (±15°) about their individual mounting points using a coupled belt and pulley arrangement, to allow the vehicle to yaw. Another group of three thrusters are mounted vertically located at intervals of 120° but aligned with the z-axis to allow vertical movement of the UDR. Therefore, the UDR is designed to be able to move laterally, heave, yaw, roll and pitch by some combination of the six axi-symmetrically installed thrusters.

Fig. 1 Hull of the UDR.

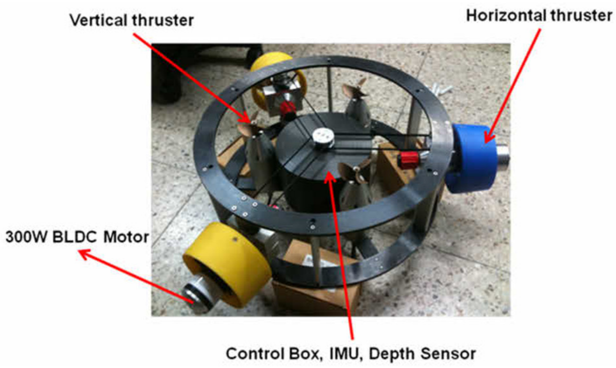

Fig. 2 Internal view of the constructed UDR.

CFD Setup for Predicting the UDR drag force

The fluid flow around the UDR has been modeled using the commercial CFD analysis code ANSYS-CFX 14.0. For these calculations, the fluid's motion is modeled using the incompressible, isothermal Reynolds-averaged-Navier-Stokes (RANS) equations in order to determine the Cartesian flow field and pressure of the water around the UDR body. The equations consist