Abstract

Improved designs for AUV hull are becoming increasingly important due to their utility in academic and industrial applications. However, a majority of such testing and design are carried out under conditions that may not reflect the operating environment of shallow water AUVs. This may lead to imprecise estimations of the AUV's performance and sub-optimal designs.

In the first stage of work experimental and numerical studies on the effect of free-stream turbulence (FST) on the evolution of flow over a cylindrical AUV hull form were carried out (it will be referred to as AUV hull hereafter) at three Reynolds numbers with different submergence depths and angles of attack. The experiments are conducted in a circulating water tank (CWC) and the instantaneous velocity profiles are recorded along the AUV hull using Acoustic Doppler Velocimetry (ADV). The experimental results of stream-wise mean velocity, turbulent kinetic energy (TKE), and Reynolds stresses are used to validate the predictive capability of a Reynolds stress model (RSM) with the wall reflection term of the pressure strain correlation. From the high fidelity RSM based simulations, it is observed that in the presence of free-stream turbulence, the pressure, skin friction, drag and lift coefficients decrease on the AUV hull. The variation of the hydrodynamic coefficients is also plotted along the AUV hull for different values of submergence depth and angles of attack with different levels of FST. The conclusions from this experimental and numerical investigation give guidance for improved design paradigms for the design of AUVs.

In the second part of the work both experimental and numerical studies carried out in conjunction, to investigate the hydrodynamic characteristics of AUV hulls at different Reynolds numbers over sloped channel-beds. The experiments have been carried out to measure the velocity field and turbulent statistics around the AUV hull with quantified uncertainty. These are contrasted against corresponding flatbed experiments to gauge the effect of testbed topography on AUV hull performance. The experimental data were used to validate RSM predictions. Hydrodynamic parameters such as drag, pressure, and skin friction coefficients were predicted from the RSM simulations at different testbed slopes, angles of attack, and drift angles of the AUV hull, to analyze the hydrodynamic performance of the AUV hull. The results presented in this study offer avenues for design improvement of AUVs operating in shallow environments,

such as the continental slope and estuaries.

In the closing part of this research, the hydrodynamic characteristics of the AUV hull are studied in rotating flow fields, which were generated in a recirculating water tank by placing a propeller in the vicinity of the AUV hull. Initially, experiments were carried out for the measurement of flow field across the AUV hull in the presence of the rotating propeller for different rotational speeds. The flow field across the AUV hull was measured using an acoustic doppler velocimeter(ADV). The measured turbulent flow statistics were used to validate the RSM-based numerical predictions in a commercial CFD solver. After preliminary validations of the turbulent flow statistics with the numerical predictions, a series of numerical simulations were performed, to study the effect of the rotational flow field of the propeller on the drag, skin friction, and pressure coefficients of the AUV hull. The operating speed and location of the propeller were also varied to check the effects on the hydrodynamic performance of the AUV hull. The results provided in this investigation will be useful for the design optimization of AUVs cruising in deeper oceans where the flow field is highly rotational because of the up welling-down welling and high eddy activities or in shallow water where the flow is highly rotational and complex because of wave-current interactions.