A CFD RANS Cavitation Prediction for Propellers

Kamal, I.M., Tuan Mohd Yusof, T.M.A

Abstract

Cavitation is a general fluid mechanics phenomenon which can occur whenever a liquid is used in a machine which induces pressure and velocity fluctuations in the fluid. The analysis of the cavitation of a screw propeller normally done in a cavitationtunnel is highly complex, difficult and time consuming. With the advances of the computer, the numerical prediction of cavitations can be done quite faster compared to the tedious manual experimentation in a cavitation tunnel. Nowadays new orientations in analysis of propeller in steady and unsteady flow are Computational Fluid Dynamics (CFD) method: using Reynolds Averaged Navier-Stokes (RANS) solver. The flow around propeller can be derived from the equations of motions using boundary conditions. Therefore the inception of the cavitation can be predicted. Therefore this paper main aim is to present the application of CFD in simulating the cavitation behaviour of ship propellers. The propeller chosen for this study is the Gawn KCA series. The code selected for the CFD in this project is FINETM/TURBO. The results of the simulated cavitation in CFDwere validated with existing experimental data. The sheet cavitations at the suction side of the propeller blade modelled by the CFD were found to be in close agreement with the existing experimental results. It was found that the CFD somehow failed to model the tip vortex cavitation occurring at the tip of the propeller blades.

Author keywords

Cavitation, CFD, Propeller, RANS