Numerical Flow Simulation around HSP Propeller in Open Water and behind a Vessel Wake Using RANS CFD Code

Authors : Kadda Boumediene, Mohamed Bouzit

Abstract : The prediction of the flow around marine propellers and vessel hulls propeller interaction is one of the challenges of Computational fluid dynamics (CFD). The CFD has emerged as a potential tool in recent years and has promising applications. The objective of the current study is to predict the hydrodynamic performances of HSP marine propeller in open water and behind a vessel. The unsteady 3-D flow was modeled numerically along with respectively the K-ω standard and K-ω SST turbulence models for steady and unsteady cases. The hydrodynamic performances such us a torque and thrust coefficients and efficiency show good agreement with the experiment results.