

## 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- $\omega$ ; standard and K- $\omega$ ; SST turbulence models for steady and unsteady cases. The hydrodynamic performances such as a torque and thrust coefficients and efficiency show good agreement with the experiment results.

**Keywords :** seiun maru propeller, steady, unstead, CFD, HSP

**Conference Title :** ICAMME 2017 : International Conference on Applied Mechanics and Mechanical Engineering

**Conference Location :** Jeddah, Saudi Arabia

**Conference Dates :** January 30-31, 2017