Abstract

The propeller is the predominant propulsion device used in ships. The performance of propeller is conventionally represented in terms of non-dimensional coefficients, i.e., thrust coefficient (KT), torque coefficient (KQ) and efficiency and their variation with advance coefficients (J). It is difficult to determine the characteristics of a full-size propeller in open water by varying the speed of the advance and the revolution rate over a range and measuring the thrust and torque of the propeller. Therefore, recourse is made to experiments with models of the propeller and the ship in which the thrust and torque of the model propeller can be conveniently measured over a range of speed of advance and revolution rate.
Experiments are very expensive and time consuming, so the present paper deals with a complete computational solution for the flow using Fluent 6.3 software. When the operating pressure was lowered below the vapor pressure of surrounding liquid it simulates cavitating condition. In the present work, Fluent 6.3 software is also used to solve advanced phenomena like cavitation of propeller. The simulation results of cavitation and open water characteristics of propeller are compared with experimental predictions, as obtained from literature [1].

References

- Bernad, S. 2006. Numerical analysis of the cavitating flows, Center of Advanced Research in Engineering Sciences, Romania Academy, Timisoara Branch, Romania.

Index Terms

Computer Science

Applied Sciences
Keywords
Propeller  CFD  Cavitation  Large Eddy Simulation  Multi phase flows  Open water characteristics

Validation