Abstract

In the present study, we conducted propeller open water test, resistance test, and propeller, ship hull interaction for a ship's resistance and propulsion performance, using computational fluid dynamics techniques, where a K-epsilon, K-omega turbulence viscous models were employed. For convenience of mesh generation, unstructured meshes were used in the propeller region of a ship, where the hull shape is formed of delicate curved surfaces. On the other hand, structured meshes were generated for the remaining part of the hull and its domain, i.e., the region of relatively simple geometry. To facilitate the rotating propeller for propeller a moving reference frame motion type technique was adopted. The computational results were validated by comparing with the existing experimental data. In this work we are interested in predicting the frictional resistance offered to a ship during its motion. To this effect we start off with a consideration of the resistance offered to the bare hull in the absence of the
propeller and later extend to the case where the propeller is in-place. The thrust generated by the propeller alone without considering the ship (called open water analysis) is also performed using CFD.

FLUENT 6.0®, was used for CFD analysis and for modeling and meshing the packages used are CATIA – V5® and ICEM-CFD® respectively. The open water analysis of the 4-bladed propeller predicted a thrust of 346 kN at 30 rps. The bare hull resistance at 228 kN at 18 m/s, and resistance with propeller in place at 18 m/s was found to be 245 kN. The results predicted by the CFD analysis were found to be suitable for the present HSDS and it is believed that the hydrodynamics design of the propeller is acceptable for the problem at hand.

Reference

- The Insean E779a Propeller Dataset, Model Geometry Description, Insean, Italian Ship Model Basin, Propulsion And Cavitation Laboratory.
- Unsteady Cavitating Turbulent Flow Simulation In A Kaplan Turbine, Shuhong Liu, Qingguang Chen, Department Of Thermal Engineering, Tsinghua University, Beijing, China, 2nd Iahr International Meeting, Timisoara, Romania, 2007.
- An Introduction To Computational Fluid Dynamics, The Finite Volume Method, By H K
- Turbulence Modeling For CFD, Second Edition By David C. Wilcox
- Fluid Dynamics Theoretical And Computational Approaches. Second Edition By Z.U.A. Warsi
- Fluent 6.2 Documentation, Users Guide.
- Principles Of Naval Architecture, Volume Ii, Edward V.Lewis.

Index Terms

Computer Science  Computational Fluid Dynamics

Key words

Resistance performance  propeller and shiphull interaction
propeller open water  Mesh
performance
sequencing and converging study
Computational fluid dynamics