Hull-Propeller Interaction and Its Effect on Propeller Cavitation

Hull-Propeller Interaction and Its Effect on Propeller Cavitation

In order to predict the required propulsion power for a ship reliably and accurately, it is not sufficient to only evaluate the resistance of the hull and the propeller performance in open water alone. Interaction effects between hull and propeller can even be a decisive factor in ship powering prediction and design optimization. The hull-propeller interaction coefficients of effective wake fraction, thrust deduction factor, and relative rotative efficiency are traditionally determined by model tests.

Self-propulsion model tests consistently show an increase in effective wake fractions when using a Kappel propeller (propellers with a tip smoothly curved towards the suction side of the blade) instead of a propeller with conventional geometry. The effective wake field, i.e. the propeller inflow when it is running behind the ship, but excluding the propeller-induced velocities, can not be measured directly and only its mean value can be determined experimentally from self-propulsion tests.

In the present work the effective wake field is computed using a hybrid simulation method, known as RANS-BEM coupling, where the flow around the ship is computed by numerically solving the Reynolds-averaged Navier–Stokes equations, while the flow around the propeller is computed by a Boundary Element Method. The velocities induced by the propeller working behind the ship are known explicitly in such method, which allows to directly compute the complete effective flow field by subtracting the induced velocities from the total velocities. This offers an opportunity for additional insight into hull-propeller interaction and the propeller's actual operating condition behind the ship, as the actual (effective) inflow is computed.

Self-propulsion simulations at model and full scale were carried out for a bulk carrier, once with a conventional propeller, and once with a Kappel propeller. However, in contrast to the experimental results, neither a significant difference in effective wake fraction nor other notable differences in effective flow were observed in the simulations. It is therefore concluded that the differences observed in model tests are not due to the different radial load distributions of the two propellers. One hypothesis is that the differences are a consequence of the geometry of the vortices shed from the propeller blades. The shape and alignment of these trailing vortices were modeled in a relatively simple way, which presumably does not reflect the differences between the propellers sufficiently.

Obtaining effective wake fields using the hybrid RANS-BEM approach at model and full scale also provides the opportunity to investigate the behind-ship cavitation performance of propellers with comparably low computational effort. The boundary element method for propeller analysis includes a partially nonlinear cavitation model, which is able to predict partial sheet cavitation and supercavitation. The cavitation behaviour of the conventional propeller and the Kappel propeller from the earlier simulations was investigated in the behind-ship condition using this model, focusing on the influence of the velocity distribution of the inflow field. Generally, the results agree well with experiments and the calculations are able to reproduce the differences between conventional and Kappel propellers seen in previous experiments. Nominal and effective wake fields at model and full scale were uniformly scaled to reach the same axial wake fraction, so that the only difference lies in the distribution of axial of velocities and in-plane velocity components. Calculations show that details of the velocity distribution have a major effect on propeller cavitation, signifying the importance of using the correct inflow, i.e. the effective wake field when evaluating propeller cavitation performance.