Propeller flow simulations in seaways including cavitation

Martin Scharf*, Moustafa Abdel-Maksoud*

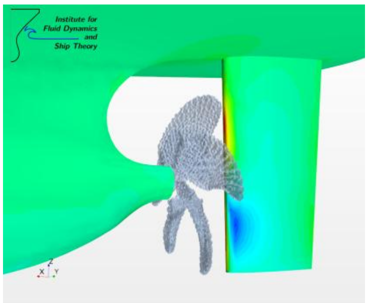
* Institute for Fluid Dynamics and Ship Theory (FDS)
Hamburg University of Technology
Am Schwarzenberg-Campus 4, 21073 Hamburg, Germany
e-mail: martin.scharf@tuhh.de, web page: https://www.tuhh.de/fds

ABSTRACT

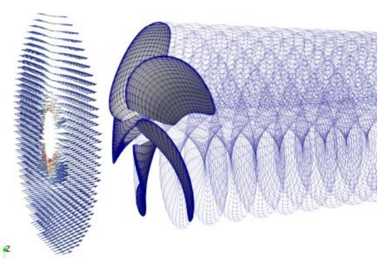
Introduction Environmental conditions have a strong influence on the propulsion performance of ships. Therefore, it is thus in the interest of researchers as well as ship owners to investigate the significant factors influencing the fuel consumption and emissions. One of the most important factors is the added resistance in waves. As finite volume methods can be applied to predict the induced resistance due to incoming waves and ship motion the integrating the propeller is often considered to be the next logical step towards simulating the whole ship in real conditions. The presence of the propeller influences the situation in many ways, e.g. increasing the hull resistance, changing the ship motions and shifting the operating point of the propeller in the unsteady wake field. Even the mean wake velocities in a temporally harmonic wake field due to body motion or wave velocities are totally different than those in the same speed in calm water due to higher order effects. Only a complete unsteady analysis is able to capture those effects without introducing large errors. Another important point is the scale effect on the hull boundary layer. It causes large differences in magnitude and characteristics between model and full scale wake fields. So while the new method need to be validated with model tests, the applicability to full scale is always important. Accurate simulations can achieved by coupling different flow methods.

Approach An efficient coupling method between the finite volume method StarCCM+ and the panel code *pan*MARE is implemented to avoid numerically expensive direct modeling of the propeller. The panel method delivers propeller forces considering the inflow extracted from the current time step of the finite volume method. These forces are then again introduced into the volume mesh of the RANS simulation to consider the interacting character of the problem. The numerical study is applied to the KCS ship which is already well investigated experimentally in regular waves. Then a mathematical model is developed to transform the obtained results for regular waves in frequency domain to the situation in a natural seaway. The influence of scale effects are examined by running simulations in model scale as well as full scale. Cavitation effects are included with a sheet cavitation model in the panel code.

Results Simulations for a ship in regular waves and forced harmonic ship motions in calm water are carried out. The recomposed first order wake fractions from the isolated simulations are already in good agreement with the extracted results from a simulation of the vessel with free motion in waves. Further evaluations will give insights into the interaction between hull and propeller, with focus on sheet cavitation.



a) representation of the propeller with active momentum source terms based on the panel forces



b) representation of the ship by import of the wake field from the RANS method into the panel code

Figure 1: Illustration of the data exchanged in the coupling method

REFERENCES

[1] B. Taskar, S. Steen and J. Eriksson, “Effect of waves on cavitation and pressure pulses of a tanker with twin podded propulsion”, *Applied Ocean Research*, Vol. **65**, pp. 206-218, (2017).

[2] P. B. Regener, Y. Mirsadraee and P. Andersen, “Nominal vs. Effective Wake Fields and Their Influence on Propeller Cavitation Performance”, *Marine Science and Engineering*, Vol. **6**, (2018)