Wake Scale Effects on Propeller Sheet Cavitation

S. Berger, M. Bauer, M. Druckenbrod, M. Abdel-Maksoud

Institute for Fluid Dynamic and Ship Theory, Hamburg University of Technology

Abstract

The present article deals with the investigation of wake scale effects on the behaviour of sheet cavitation and propeller-induced pressure fluctuations.

For this purpose, a numerical method is introduced, which is based on coupling a viscous flow solver with a potential flow theory-based boundary element method. The viscous flow solver is used to calculate the effective wake field of the ship. The wake field is then used as input for the in-house BEM code *pan*MARE determining the pressure distribution on the propeller blades. In return, the BEM code passes a corresponding body force distribution to the viscous flow solver in order to simulate the impact of the propeller on the flow around the aftship. Furthermore, an unsteady partial sheet cavitation model has been developed and implemented in the BEM code.

For the numerical case study carried out in this work, the KRISO Container Ship is chosen as example. The full-scale ship and the model are investigated with respect to pressure fluctuations. The results of the simulations are compared to results obtained by experiments in a cavitation tunnel.

The implemented sheet cavitation model allows looking in detail at the extent and the time-dependent volume of the cavity sheet. Thus, it is possible to point out the differences between the behaviour of the cavity sheet of the full-scale ship and the model.

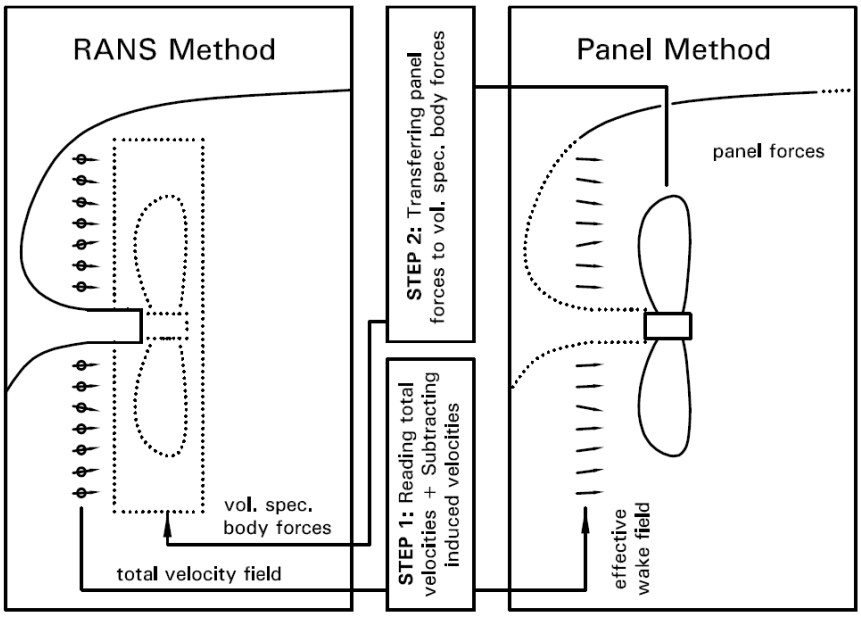

Figure 1: Principle of the coupling procedure.

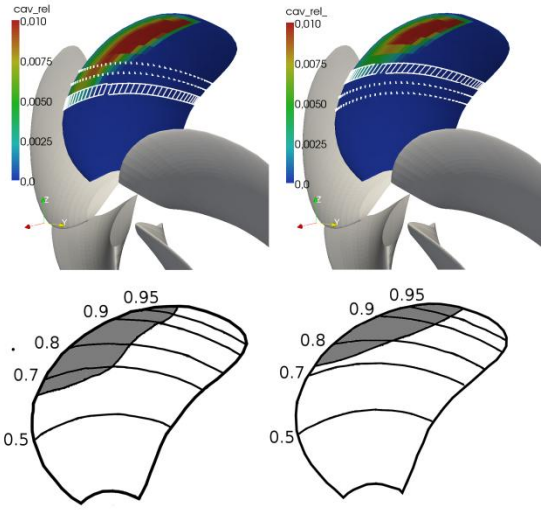

Figure 2: Predicted and experimentally found cavity sheet extent for the full-scale ship and the model.

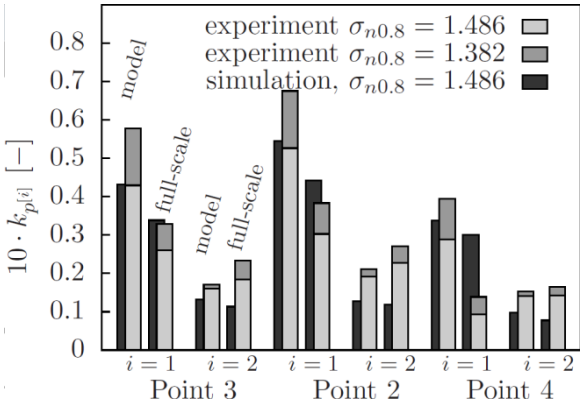

Figure 3: Predicted and experimentally found pressure pulses for the full-scale ship and the model.

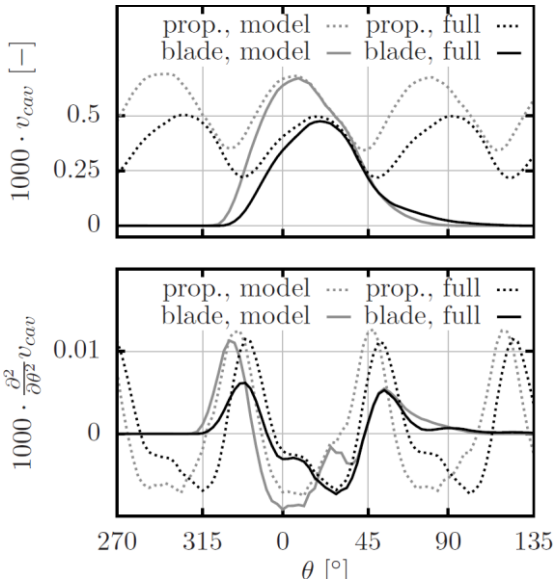

Figure 4: Cavity volume and cavity volume acceleration.

Hinweis: Hochauflösende Bilder und Diagramme können wir jederzeit nachliefern.