Simplified CFD Approach for Simulating Propeller Cavitation in Ship Wake Fields

Keun Woo Shin[1] (keun.shin@man-es.com) and Poul Andersen[2] (pa@mek.dtu.dk)

[1]Propeller & Aftship R&D Department, MAN Energy Solutions, Frederikshavn, Denmark
[2]Department of Mechanical Engineering (MEK), Technical University of Denmark (DTU), Kgs. Lyngby, Denmark

CFD made by a turbulent viscous flow solver is generally known to have higher accuracy in simulating unsteady cavitation on ship propellers than inviscid flow solvers. The increase of computational capacity enables practical use of CFD methods for propeller cavitation simulations, but it is still CPU-intensive for repetitive simulations in propeller design process.

Computational effort can be reduced by modelling non-uniform hull wake numerically as a propeller inflow instead of including a hull model (Shin et al 2011), but it still requires considerable computational effort, because the refinement of spatial and temporal discretization is necessary for resolving cavitation variation and it needs numerous iterations to reach a periodically converged solution (Shin & Andersen 2018a).

A 4-blade propeller designed for a single-screw 35,000 DWT bulk carrier is considered as a test case. The result of a cavitation tunnel test made on the propeller with the whole hull model shows extensive sheet cavitation. Cavitation simulations made by DES with a full non-uniform hull wake model have been validated against the experimental result regarding sheet cavitation variation and detachment (Shin et al 2015).

Hull wake is simplified to be circumferentially uniform and to vary only along the radial direction for representing a specific blade position (Shin & Andersen 2018b). Quasi-steady cavitation simulations are made only on a single blade with periodic boundary conditions and moving reference frame in a quarter of a cylindrical fluid domain.

The possibility of cavitation simulations with simplified hull wake models for reflecting cavitation patterns at characteristic blade positions showing the maximum extent and detachment of sheet cavity is examined by investigating the correlation between the results of CFD with full and simplified hull wake models. The distributions of the vapor volume fraction and pressure coefficient on constant-radius sections are compared. The fluctuations of vapor volume and pressure at the cavity closure region are also investigated for predicting erosion risk at blade positions showing dynamic cavity collapse.

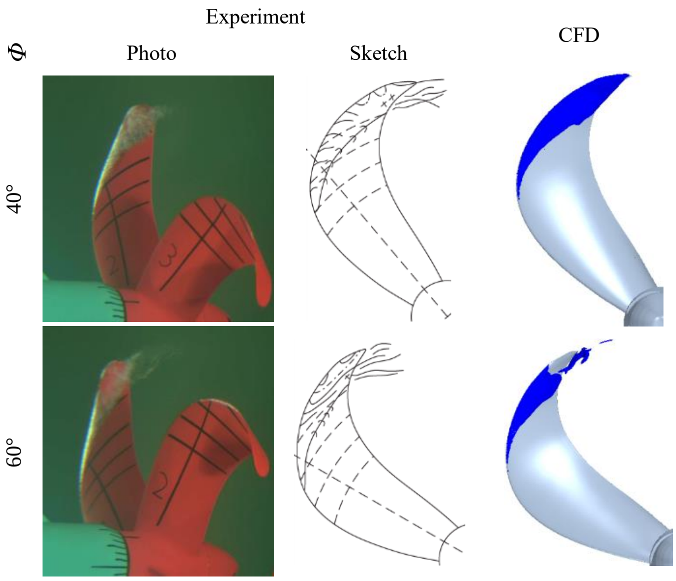

Figure 1: Unsteady cavitation from experiment and CFD (Shin et al 2015)

References

Shin, K. W., Andersen, P. & Mikkelsen, R. (2011). ‘Cavitation simulation on conventional and highly-skewed propellers in the behind-hull condition’. Proc. of SMP’11, Hamburg, Germany.

Shin, K.W., Regener, P.B. & Andersen, P. (2015). Methods for cavitation prediction on tip-modified propellers in ship wake fields. Proc. of SMP’15, Austin, Texas, USA

Shin, K. W. & Andersen, P. (2018a). ‘Numerical study on characteristics of cloud cavitation on a ship propeller’. Proc. of 3rd International Meeting on Propeller Cavitation, Istanbul, Turkey

Shin, K.W. & Andersen, P. (2018b). ‘Simplified CFD Approach for Simulating Propeller Flows in Ship Wake Fields’. Proc. of NuTTS’18, Cortona, Italy