CFD Analysis of Turning Circles in Regular Waves for ONR Tumblehome

MARINE 2023

Shawn Aram* and Philipp Mucha†

* Naval Surface Warfare Center Carderock Division (NSWCCD)
9500 MacArthur Blvd.
West Bethesda, MD, USA
e-mail: shawn.aram.civ@us.navy.mil, web page: https://www.navsea.navy.mil/

† Siemens Digital Industries Software
200 5th Avenue
Waltham, MA 01720
e-mail: philipp.mucha@siemens.com, web page: http://www.sw.siemens.com

ABSTRACT

Maneuvering in waves (MIV) has to be accounted for in the design phase to ensure propulsion and steering device can enable ships to withstand adverse conditions and sustain sufficient maneuverability. As part of the ship stability and maneuverability assessment in heavy weather, the safety of the crew, payload and vessel is of concern and sought to be guaranteed by regulatory frameworks of the International Maritime Organization (IMO). Despite the importance of maneuvering in waves and new rules set forth by the Maritime Environmental Protection Committee (MEPC), the majority of the ship hydrodynamic research has been only focused on operations in calm water. Reported experimental studies are very limited to support the ship design or inform numerical validation studies. A Reliable numerical modelling of free-running maneuvers is challenging, as it requires accurate predictions of (a) flow features such as cross-flow and bluff body separations around the ship, (b) incident waves and their interactions with the ship, (c) propulsion system, and (d) hull-propeller-rudder interactions.

Numerical methods with a wide range of fidelity levels have been developed to study ship maneuvering in waves. Among them is Computational Fluid Dynamics (CFD) which has gained attention in recent years through enhancement in numerical methods and increase in computational power, and could be used to understand detailed flow physics governing the MIW problem, support the development of reduced-order models (ROM), and guide experiments. Recent analysis by Aram and Wundrow [1] showed promising results for the application of CFD for improving an approach based on potential flow theory with a significantly shorter time to solution (~ 200-fold speed-up). The analysis also indicated ample room for improving accuracy of the numerical models.

One of the contributors to the discrepancy between the reported CFD and model test data in the Aram and Wundrow [1] analysis was believed to be the body force based propeller model adopted in the study. This paper will be a continuation of Aram and Wundrow's study to investigate the effects of the propeller modelling choice in CFD for maneuvering prediction of a ship in waves. Two propeller models including a body force model and a discretized propeller model are used to predict a turning circle maneuver of the Office of Naval Research (ONR) Tumblehome ship in regular waves at 35° rudder angle. The maneuvering characteristics prediction of the ship is validated against the model test data by Sanada et al. [2] at three wave lengths of $\lambda/L = 0.5$, 1.0, and 1.2. Simulations are also performed to study the effects of wave length and steepness (h/λ) on the maneuvering performance of the ship.

REFERENCES

[1] S. Aram, S. and D. Wundrow, "Application of blended-method and CFD to ship maneuvering prediction", 34th Symposium on Naval Hydrodynamics, 26 June – 1 July 2022, Washington, DC, USA.

[2] Y. Sanada, H. Elshiekh, Y. Toda, and F. Stern, "ONR tumblehome course keeping and maneuvering in calm water and waves," *Journal of Marine Science and Technology*, 24(3), (2018).