XI International Conference on Computational Methods in Marine Engineering

High-fidelity Simulation of the Turbulence Flow Around a Rotor Sail

Kyobin Lee[1*], Sanghyeon Kim[1] and Hyoungsuk Lee[1]

[1] Hyundai Maritime Research Institute,
HD Hyundai Heavy Industries, 44032 Ulsan, Korea.

[*] kyobinlee@hd.com

ABSTRACT

This study investigates the turbulent flow around a rotor sail through high-fidelity numerical simulations. Key flow phenomena, such as boundary layer separation, vortex shedding, and wake stability, were analyzed.

The simulations are validated against experimental data conducted by Bordogna, G. to assess their accuracy in predicting aerodynamic performance, including lift and drag forces.

Numerical simulation shows significant discrepancies in lift predictions depending on the turbulence models. The IDDES model more accurately captured unsteady vortex structures and turbulence interactions in the wake, while the SST model showed limitations, particularly at high rotational speeds, resulting in greater deviations. In contrast, drag predictions showed good agreement with experimental data across both turbulence models.

The findings emphasize the critical influence of turbulence model on capturing the complex interactions between flow structures and the Magnus effect.

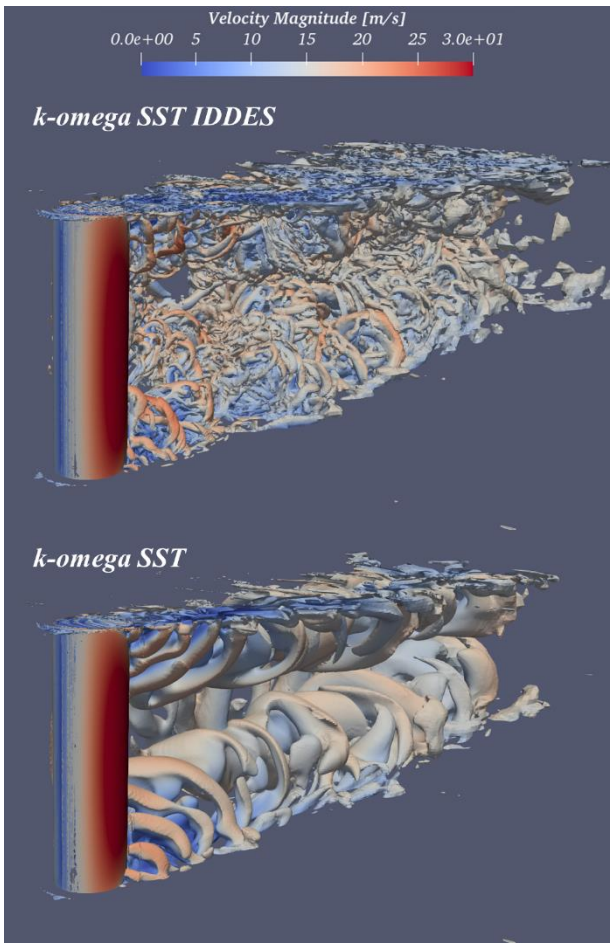

Figure 1: Turbulence structure around a rotor sail

References

Bordogna, G. (2020). Aerodynamics of wind-assisted ships: Interaction effects on the aerodynamic performance of multiple wind-propulsion systems. https://doi.org/10.4233/uuid:96eda9cd-3163-4c6b-9b9f-e9fa329df071.