Performance Modelling of Ship Propellers by Computational Flow Simulation

Bernhard Semlitsch

Institute for Energy Systems and Thermodynamics, TU Wien, Getreidemarkt 9, 1060 Wien, Austria

Climate change causes more extreme weather conditions, i.e. hotter summers and colder winters, and less distributed rain. That has consequences on the amount of water carried in rivers. Because of the shallow water levels occurring during longer-lasting dry seasons, the draft of ships must be restricted to prevent damage to the riverbed. Under such conditions, cargo ships are forced to limit the load below their capacity. The reduction of inland shipping has wide-ranging consequences because some ships can transport 3,000 tons of cargo, equivalent to 150 trucks. Road transportation generates more CO_2eq emissions than inland shipping [1]. Therefore, inland vessels with a lower draft are in demand, which can continue the undisturbed transport even in low water periods.

One of the determining factors for the draft of the ship is its propeller, which has to be fully submerged and capable of pushing the vessel with a certain thrust. The thrust is proportional to the quadratic forward velocity of ships, the density of the river water, and the propeller area. Hence, the propellers are designed with the largest possible cross-section because the ship shall reach its target as fast as possible. The diameter of the propeller defines, therefore, the draft of inland vessels. This project aims to replace the large propeller with two smaller propellers while the thrust and the total propeller area are preserved. Thereby, the ship's draft can be reduced under the challenge of providing the same thrust generated by a single propeller. The two propellers are arranged close to each other, enabling the utilisation of the swirling energy that is lost with single propellers. The generated swirl component of the individual propellers shall be converted into thrust using the fluid interactions and a surrounding nozzle.

Numerical flow simulations are employed to find the optimal propeller arrangement and nozzle shape, reducing the ship's draft. The simulation methodology of the flow phenomena occurring around the propeller must correctly replicate turbulent flow features and cavitation in a multiphase environment. Further, the rotation of the propellers needs to be modelled with respect to the static geometries. The close arrangement of the propellers represents the challenge for the numerical mesh generation because the conventional sliding-mesh approach cannot handle such tight spacings. Another methodology is the overset grid technique [2], where the flow quantities are computed on overlapping meshes moving independently on top of each other. The connection between the grids is built by interpolating the cell and point values. The steep pressure gradients occurring due to cavitation and the turbulent flow demand a fine mesh resolution. Therefore, the governing equations, i.e. the Navier-Stokes equations, are numerically solved with high accuracy on fine overset grids to meet the associated requirements.

The application of such numerical shape optimisations for the industrial design of ship propellers is presented. The computational requirements of the individual numerical models and assumptions are stated and analysed. The need for high-performance computers is discussed when the simulation complexity of optimisation steps considering the correct physical representation is as high as for ship propeller design.

We acknowledge funding as a PRACE SHAPE project by the EU's Horizon 2020 research and innovation programme (2014-2020) under grant agreement 823767. The work is achieved using the resources of VSC-4 provided by the Vienna Scientific Cluster (VSC).

References

[1] Van Fan, Y., Perry, S., Klemeš, J.J. and Lee, C.T., J. Clean. Prod., **194**, 673 (2018)

[2] Tang, H. S., Jones, S. C., and Sotiropoulos, F., J. Comput. Phys., **191**(2), 567 (2003)