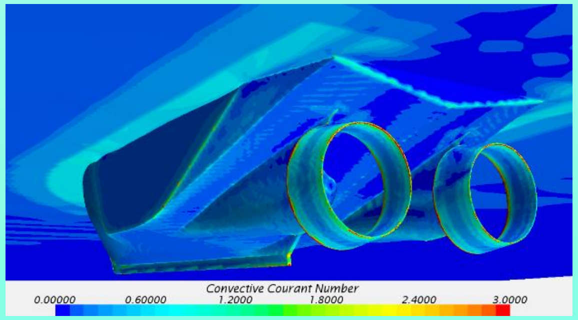
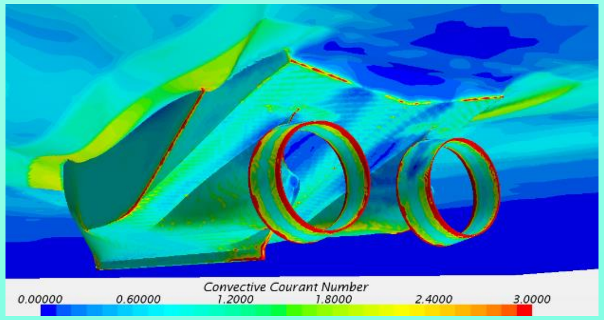
Investigation of Hydrodynamic Performance of a Twin-Screw Trawler Using CFD

Norwegian University of Science and Technology

Author: Nikolas Øksdal Svoren
E-mail: nikolasv@stud.ntnu.no
Supervisor: Sverre Steen
Companies:



4 knot simulation



10 knot simulation

Scope of Work

As a part of a environmental research project the ship design consultants Seacon have designed a twin-screw trawler, with large ducted propellers and steerable ducts. From the project thesis a new hull geometry design was proposed. The objective of this master thesis is to verify the hydrodynamic performance and compare the two designs.

For the master thesis, the following issues and activities shall be covered:

- Give an introduction to the proposed trawler design, and how it stands out from previous design and typical trawler designs, with focus on hydrodynamic aspects.
- Discuss potential hydrodynamics-related problems of the current design, and what can be done to address these problems.
- Perform a resistance and flow prediction of the current design in straight-ahead cruising condition. Perform validation and convergence study.
- Special emphasis shall be put on the inflow conditions for the propeller. Derive the nominal wake in straight-ahead at different speeds. Consider the effect of the suction of the propeller (effective wake).
- Give recommendations for improvements of the hull design.

Method

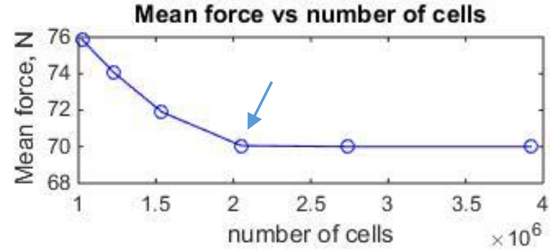
STAR CCM+ was chosen as CFD software to analyze two hull designs. After setting up a initial simulation with suitable physical models, boundary conditions, and volumetric controls, a convergence study is to be preformed. This is to validate and find the correct mesh and domain size as well as a good time step to obtain a solution with reasonable wall

Preliminary Results

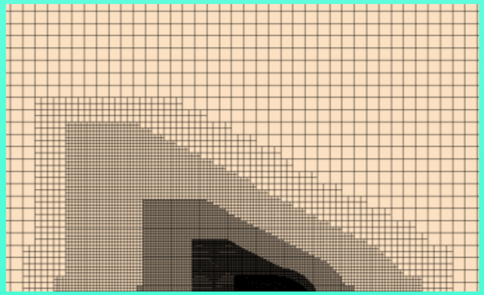
Mesh

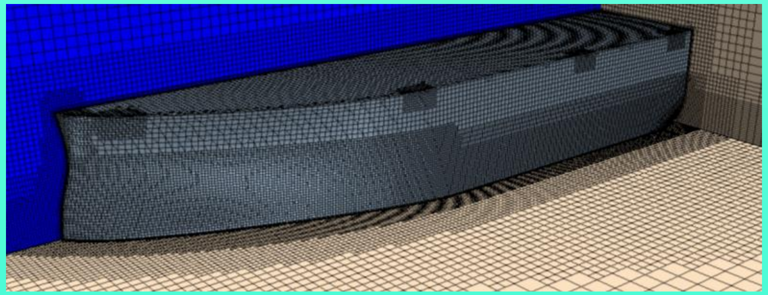
A total of six different mesh refinements was tested ranging from 1 mil cells to 4 mil cells. A clear convergence was found as seen in figure below.

Base value	Interval	Difference [%]
0.24 – 0.22	1	2.386
0.22 – 0.20	2	2.937
0.20 – 0.18	3	2.563
0.18 – 0.16	4	0.077
0.16 – 0.14	5	0.004



Base value of 0.18m with 2mil cells was chosen for further analysis, seen in figure below:

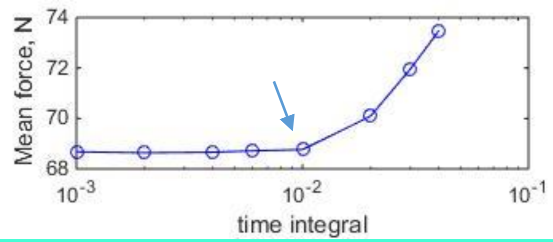




Time

Eight different time steps was simulated. These simulations also converged nicely, and a time step of 0.01s was chosen.

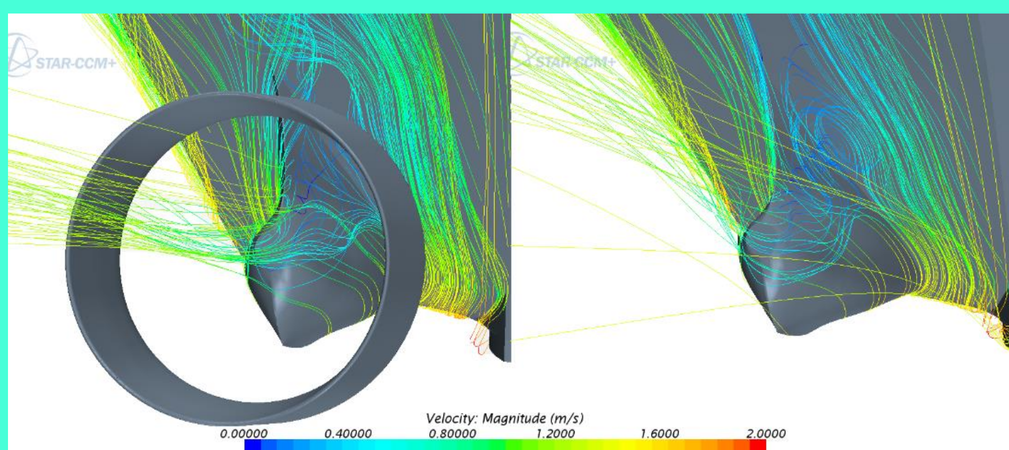
Time step	Interval	Difference [%]
0.001-0.002	1	0.156
0.002-0.004	2	0.033
0.004-0.006	3	0.081
0.006-0.01	4	0.072
0.01-0.02	5	1.889
0.02-0.03	6	2.597
0.03-0.04	7	2.047

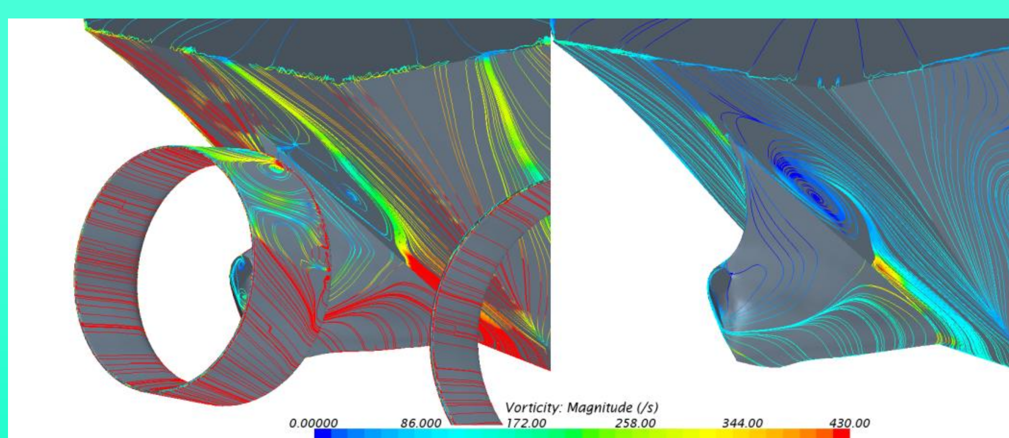


y+ value and CFL number. The convergence test are based upon methods form TMR 15 Numerical Methods. MatLab was used to post process results, and plotting. Results from this test was then used to set up a final simulation. A total of 8 simulations was then created, four of each type of hull, two simulations on 4 knots and two with 10 knot(full scale). Finally each of these setups was set up with and without nozzle and actuator disk to simulate both nominal and effective wake.

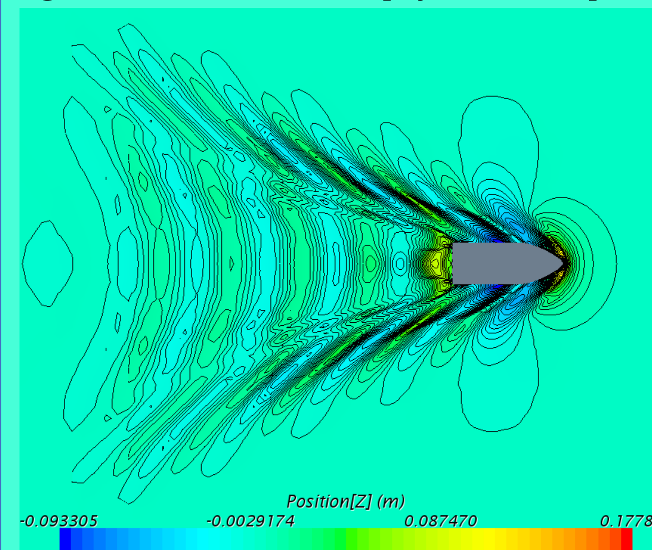
Preliminary comparison

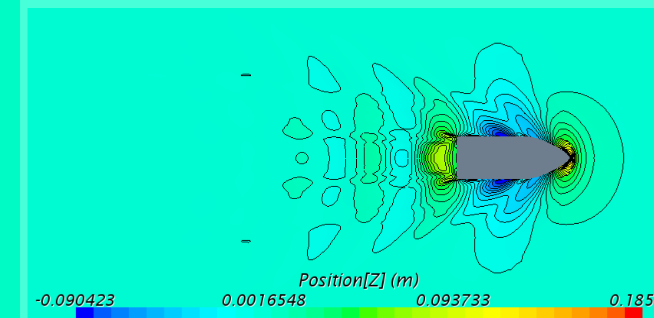
This poster contains only preliminary results and comparisons since all simulations are not yet completed. From the figures below one can see some differences between one model with and without nozzle and actuator disk at 10 knot full scale speed. The first picture display streamlines where the color identify the velocity of that specific streamline. The second figure display water particle path, where the color display the vorticity. Note the read lines display 430 rotations /s and up.





The time step directly influence amount of calculations preformed and therefor length of the simulation time. It also influence quality of the simulation, example being details in the wave elevation as seen below. Figure on the left is a simulation with time step of 0.01s, while on the right is an identical simulation, but with a time step of 0.08s. The figure on left clearly shows the classical Kelvin wake pattern which is expected from a moving ship, while on the right one can observe a unphysical wave pattern for a ship with Froude number of 0.33.

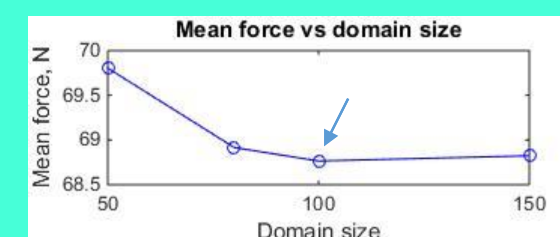




Domain

The domain size is the volume surrounding the hull which is include in the calculations. Domain used in the previous simulations was set to 4lwl back and forth, 1.7lwl up, 5lwl to the side and 3.5lwl down. The other domains tested was 50% , 80% and 150% of this setup. It was decided upon sticking with the domain used in the previous simulations.

Domain size	Interval	Difference [%]
50% - 80%	1	1.288
80% - 100%	2	0.219
100% - 150%	3	0.085



Most important references:

- CD-Adapco. (2014). STAR CCM+ User Guide.
- Cengel, Y., & Cimbala, J. (2010). Fluid Mechanics Fluid Mechanics (Vol. 2, pp. 853-920): Mc Graw Hill.
- Steen, S. (2007). Motstand og propulsjon propell og foil teori (Vol. 2). Institutt for Marin Teknikk.