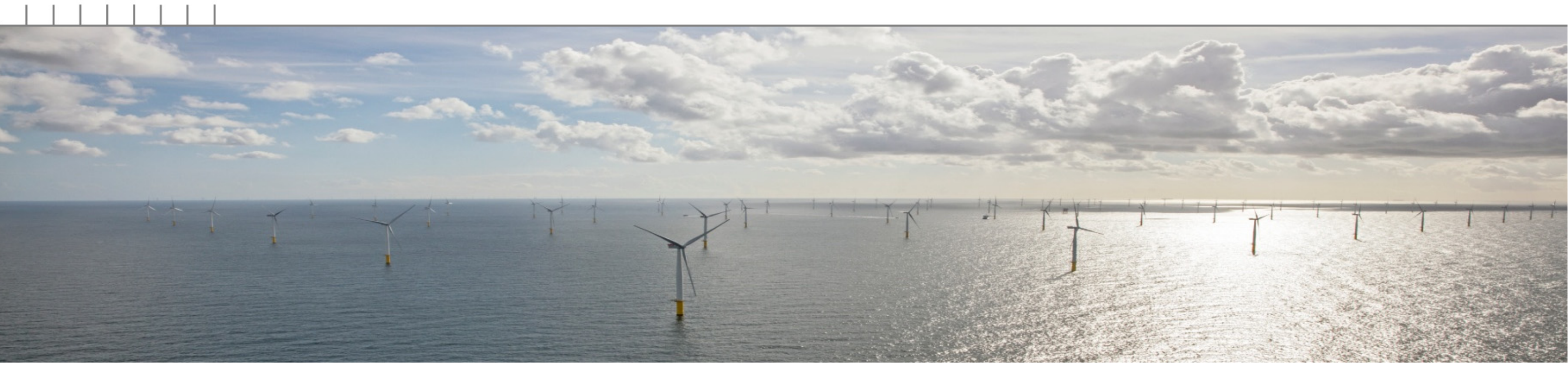

CFD for Thruster Interaction

To determine the Dynamic Positioning (DP) capability of vessels and semi-submersibles at sea, input for wind, current and wave loads are required, as well as the available thrust, thruster layout and thruster-interaction losses. The effective force generated by the thrusters during a DP operation can be significantly smaller than what would be expected based on the thrusters' open water characteristics. With CFD calculations, it is possible to investigate the thruster losses.

CFD simulations can be used to determine:

- Thruster – hull interaction;
- Thruster – thruster interaction;
- Thruster – current interaction;
- Confined water effects.

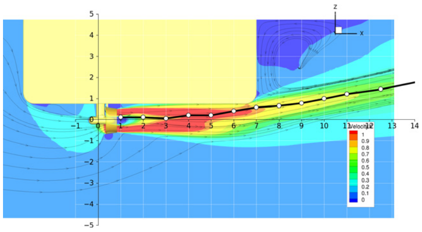

Figure 1: Thruster wake comparison between PIV experiments (black line) and the velocity field obtained from CFD (colour) for a thruster under a barge.

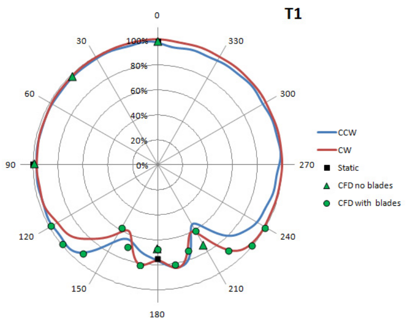

Figure 2: Thruster-hull interaction comparison between experiments (lines) and CFD (symbols) on a drill ship for which one thruster is active.

CFD philosophy at MARIN

MARIN has been developing viscous flow CFD codes since the beginning of the 1990's. The philosophy behind our own CFD code ReFRESCO is "Reliable", "Fast" and dedicated to "ships and offshore constructions". This is reflected in robust developments, and combining proven technology with new trends. Special attention is paid to accuracy within all ReFRESCO developments: code verification, solution verification and solution validation is done for any new application. In order to perform large calculations, MARIN has its own cluster, with 4000 cores available.

Thruster – hull interaction

Due to the thruster blade rotation, a flow is developing in the wake of the thruster that flows along, and interacts with the hull. This is denoted as thruster-hull interaction, and reduces the effective thrust of the thrusters. Furthermore, the Coanda effect is deflecting the wake of the thruster upwards, presented in Figure 1 for a thruster under a barge. The black line denotes PIV experimental data of the wake path, CFD results are presented with the colour contours of the velocity. The path of the wake is accurately captured by the CFD results.

Figure 2 presents results for thruster-hull interaction for a drillship with six thrusters, of which one thruster is active. The solid lines show the experimental data, where the thruster is rotated either in clockwise (CW) or counter clockwise (CCW) direction. The symbols denote the CFD results. Good agreement for the thruster-interaction (within 5% of the experiments) is obtained between experiments and CFD simulations.

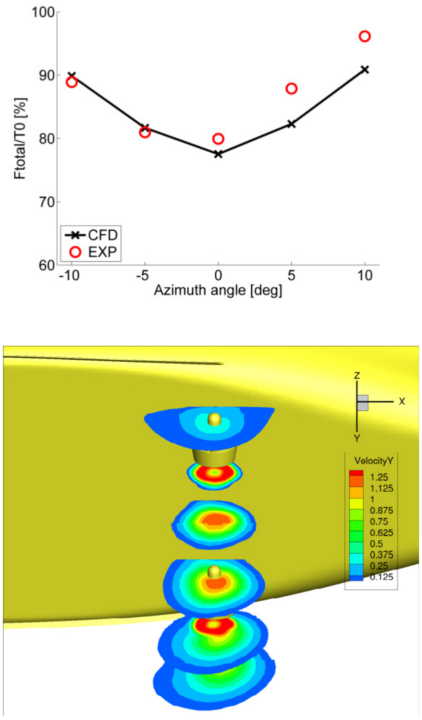

Figure 3: Thruster-thruster interaction for different CFD methods compared to experimental data (top) and the stream wise velocity for one thruster angle.

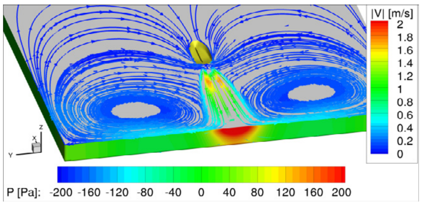

Figure 4: Flow at the free surface due to the thrusters in a confined water area, and the pressure on the quay. The flow shows two clear recirculation areas.

Thruster – thruster interaction

Thruster-thruster interaction can lead to a large drop in thruster efficiency, which impacts the station keeping capabilities of the vessel. Results of thruster-thruster interaction simulations are presented in Figure 3, for which one thruster is aimed at the other thruster. For the worst case scenario where two thrusters are directly aimed at each other, there is a thrust loss of more than 20%. In Figure 3, it is shown that the CFD predictions capture the trend observed in experiments within 5% for a range of azimuth angles of the upstream thruster.
The velocity in the wake of the thruster is illustrated in the bottom of Figure 3.

Thruster efficiency in confined water

Bollard pull trials are ideally performed in unrestricted water conditions, in which there is no influence of shallow water or a nearby quay. In reality, this is hard to achieve. CFD can be used to determine the confined water effects, and correct for these to obtain the unrestricted bollard pull performance. Figure 4 shows an example of bollard pull conditions in a test basin for very confined water. Due to the flow induced by the thrusters, a flow recirculation is formed inside the basin, which reduces the thruster efficiency by up to 8%.

ReFRESCO Operation

Are you interested in performing your own CFD computations, or would you like to collaborate with MARIN? Our CFD code ReFRESCO is available to you through ReFRESCO Operation. Please contact refresco@marin.nl for further information.

For more information contact MARIN:
Patrick Schrijvers
T +31 317 49 35 96
E p.schrijvers@marin.nl

Arjen Koop
T +31 317 49 32 68
E a.koop@marin.nl