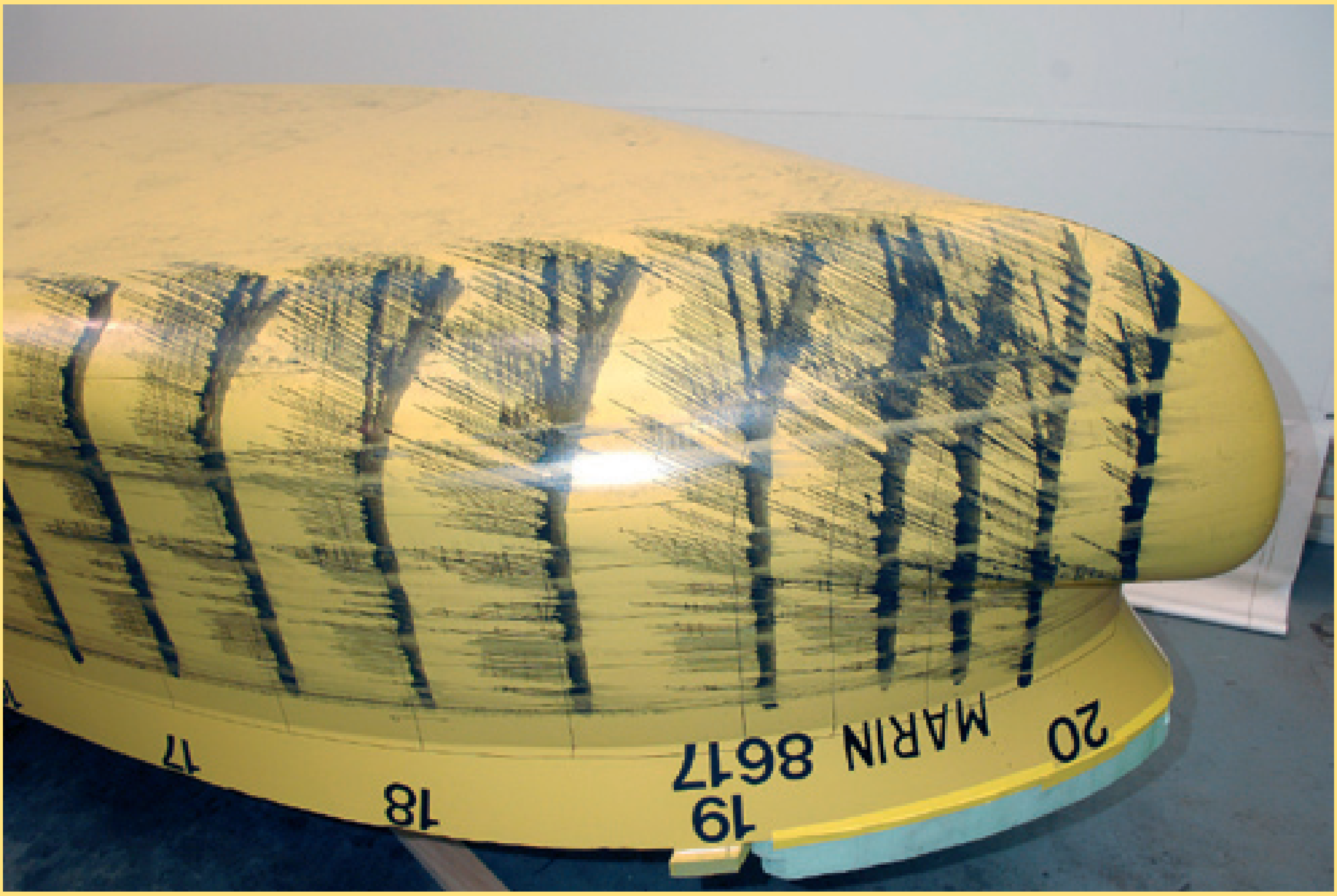
MARIN's CFD simulation proves to be a numerical alternative for the flow-aligned orientation of hull appendages

Martin Hoekstra &
Bram Starke
M.Hoekstra@marin.nl

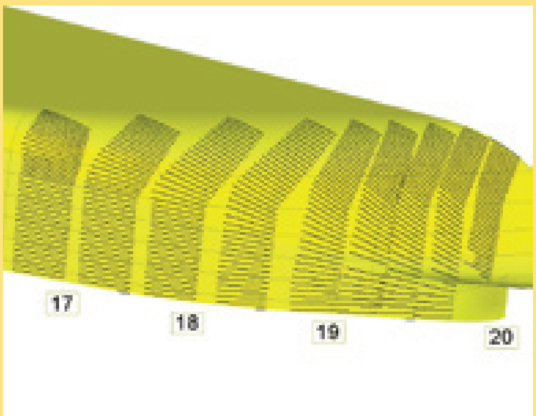
CFD takes charge



56880

Computational Fluid Dynamics (CFD) is the common denominator of various kinds of numerical flow simulations, with ship hull form optimisation the most developed application. Recently, the viscous flow solver PARNASSOS, has been validated for flow alignment of appendages, notably bilge keels, struts, scallops and the like. A powerful numerical application has been added to MARIN's toolbox for ship design support. Report explains.

Example of limiting streamlines on the hull in an experiment (top) and a representation of the numerical equivalent (bottom).



The proper orientation of shaft-supporting struts or of scallops and grids at thruster-tunnel openings, as well as the positioning of bilge keels, is a recurring issue for several ship types. Flow line tracing by paint-flow or tuft tests is the traditional approach in providing information for good advice. These tests seem simple and straightforward but are often cumbersome, especially in a towing tank, as opposed to a water or wind tunnel. They may require several iterations in order not to produce intolerable uncertainty into the interpretation of the results. In addition, paint tests pollute the tank water.

Complete flow field

For the positioning of bilge keels paint-flow tests give adequate information because there is little cross-flow (variation of flow direction through the boundary layer) locally. Bow-thruster openings, however, are found quite close to the bow where the three-dimensionality of the flow is strong. Visualisation of the direction of the skin friction on the hull surface may not be representative then for the flow direction a little distance away from the hull.

A clear advantage of the alternative CFD route - in MARIN's case applying the viscous-flow solver PARNASSOS - is that information on the complete flow field is obtained. This provides better advice to customers. As an example, the figure shows the results of a paint test and the equivalent numerical result.

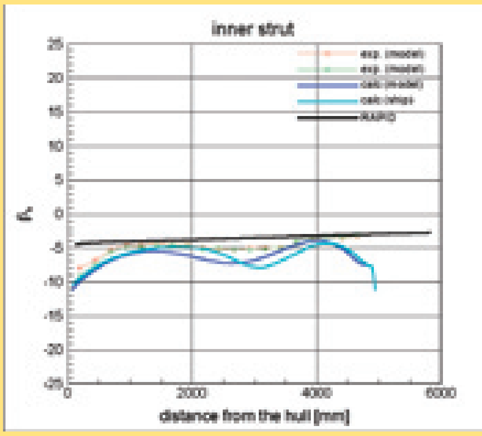
For twin-screw ships with open shafts, the flow-aligned orientation of the struts supporting the shaft is relevant and even of crucial importance for high advance speeds when there is a risk of cavitation occurring on the struts. In span-wise direction these struts extend from the hull surface to outside the hull boundary layer and can experience an appreciable variation in local flow orientation. In addition, partly uncovered shafts bring the surrounding fluid in rotation and can cause extra complications at the strut end fitted to the barrel containing the shaft gearing.

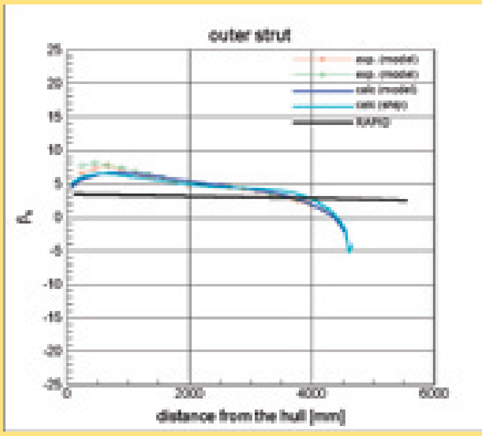
Good numerical alternative

Provided that free surface deformation, sinkage and trim are taken into account, CFD can give a detailed picture of the flow in the vicinity of a strut. It can also give guidance in the twisting of the strut, or - if twist is to be avoided for ease of construction – in choosing a proper mean orientation angle. The influence of the propeller on the flow direction at the position of the struts can readily be included.

Of course, several correlation studies have been made between experimental and computational results to corroborate the reliability of the numerical predictions. The figure shows the local flow inclination angle in the horizontal plane for the two struts of a V-bracket configuration. These results include two sets of experimental data, predictions by the inviscid flow code RAPID and the viscous flow code PARNASSOS, (the latter with and without the effect of the propeller). The PARNASSOS results compare quite well with the experiments.

In conclusion, there is a good numerical alternative now for the flow-aligned orientation of hull appendages.





Comparison of results of measurement and computation of the flow orientation angle in horizontal planes for a V-strut configuration (symbols: experiment, black line: potential flow computation, read line: viscous flow computation without propeller action, green line: viscous flow computation with propeller action).