Predicting the flow

For years hydrodynamicists have been trying to mathematically simulate the three-dimensional (3-D) flow that a vessel experiences underway. Scott C. McClure, president and Donald D. Burris, chief hydrodynamicist, Alan C. McClure Associates explain the challenges

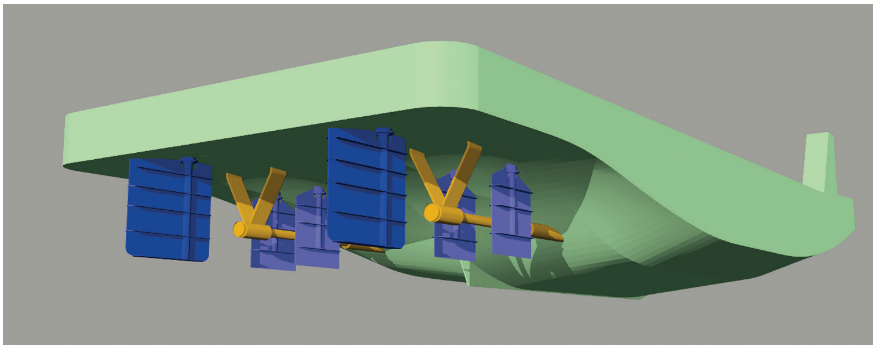


Figure 1. Overall geometry of study vessel

Initially plagued by the high cost of computational power, as well as crude software, it appeared this goal would only be accomplished in the distant future. However, with the recent dramatic advances in high-speed computers, today's computational fluid dynamics (CFD) software packages enable credible and accurate results on a cost-effective scale.

In late 2008, Alan C. McClure Associates, Inc. (ACMA) decided to make a significant investment in computer resources and in the STAR-CCM+ software developed by CD-Adapco. This software allows ACMA to perform CFD analysis that breaks a 3-D volume of space into a number of volumetric cells. Mathematical equations based on the various physics models are then calculated for each of the cells in the volume, with the solution of one cell setting the boundary conditions of adjacent cells. Changes to the solution of the physics equations over time are then calculated to simulate real-world effects.

ACMA recently performed a propulsion study to simulate the hydrodynamic effects that a vessel would experience while underway. The study was done in three phases: a base flow model, momentum source model, and a fully modelled rotating propeller. The base model provided an understanding of the simple flow around the hull and appendages - hull, rudders (steering and flanking), propeller shaft and support struts. The base model was then modified to include a momentum source disk to simulate a four-blade propeller. A final model phase utilised a 3-D model of a rotating five-blade propeller.

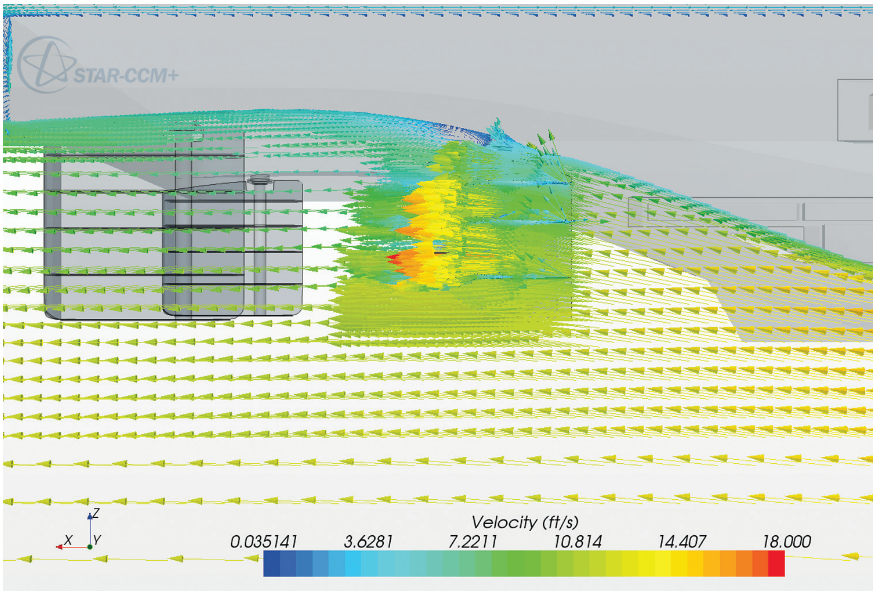
Basic flow model

The basic flow model was completed in order to observe flow phenomena around the hull in the absence of the propeller. This model was invaluable in attempts to identify areas of potential separation of flow from the hull, as well as areas of turbulent flow that would be exaggerated by a propeller. It was observed that there was a significant transverse flow that initiated from the chine into the outboard flanking rudders, imparting flow disturbances into the propeller. This transverse flow would tend to increase the likelihood of the vessel developing vibration issues that could lead to premature steel component and equipment failures, as well as making a noisy, uncomfortable ride for the crew.

Although there was a lot of transverse flow as the water came off the chine, there was not any separation of the flow as it moved up the buttocks longitudinally. Separation of the flow due to steepness of the buttock lines is frequently a concern since owners/naval architects want more power in a compact hull form leading to steeper buttock lines to accommodate propulsion equipment.

As the power requirement goes up, the ability of the propeller to absorb this additional power requirement means that the blade area ratio (BAR) needs to increase and/or the wheel needs to have a

Figure 2. Basic flow model, flow on outboard flanking rudder



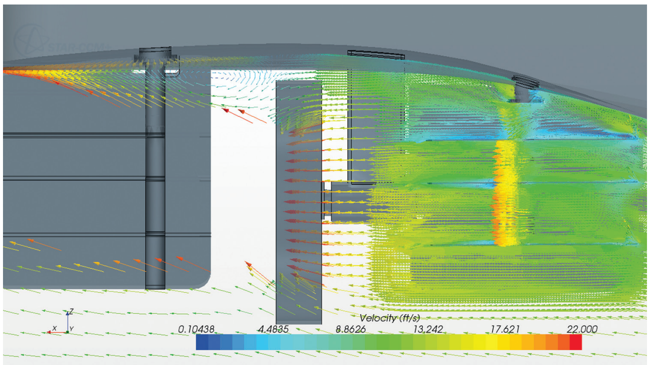


Figure 3. Momentum source propeller model

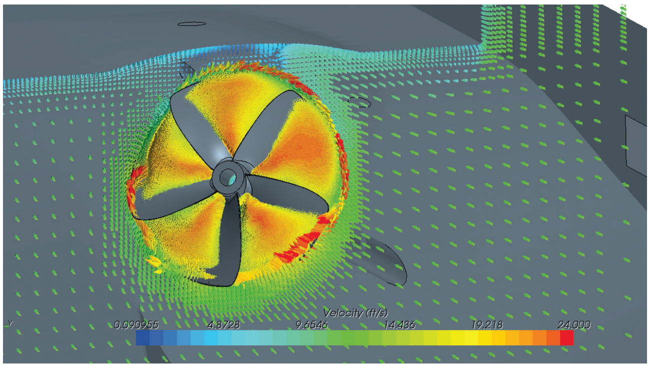


Figure 4. Rotating propeller model

larger diameter. As the propeller diameter increases, the clearance between the propeller tips and the hull, as well as the submergence of the wheel often suffer. These trade-off's and decisions are always present when the hull characteristics remain constant and the need for more power is desired.

The momentum source disk model

The next phase of the modelling used a momentum source to simulate a rotating propeller. The momentum source is based on fan laws, and thus the performance curve of a four-blade propeller was input to the model. As flow moves into the vicinity of the momentum source, the flow is accelerated as if it were being "sucked" and "pushed" by rotating blades. The four blade propeller was used as it is a common selection for vessels of the type and size under consideration. Although we noted that the 90degs blade orientation might create problematic interactions with the flanking rudder and shaft struts, ACMA felt that the large BAR of the wheel might mitigate those issues.

Based on real-time monitoring of the analysis, 130 seconds of simulation time was found sufficient to quantify the steady state flow phenomena. A time-step of 0.05 seconds was used for the simulation, based on the blade passing frequency. Based on the physics models used and the density of the volume mesh, each time-step represented roughly 64 million simultaneous equations being solved.

The resulting flow field revealed flow lines at the top of the propeller moving much slower than the flow velocity at the bottom of the propeller. This type of asymmetrical loading tends to accentuate any vibration problems such as those already noted due to the large transverse flow. What was also noted was that a force coupling developed with the impact loads on the hull at the blade passing frequency. This coupling was due in part to the proximity of the propeller blades to the hull.

The five-blade rotating propeller model

A number of options to mitigate the adverse effects that were seen with the four-blade propeller were proposed, including a five blade propeller. This model also stepped to a further level of complexity by including a 3-D rotating model of the five blade propeller. In this simulation, the solid representing the propeller was rotated in a fluid domain embedded within the larger fluid domain. The software mapped the physics effects between the two domains such that the flow was accelerated into the propeller and thrust aft.

The results were promising. The flow irregularities were somewhat tempered owing to the increased passing frequency of the blades. The change in propeller geometry lead to de-coupling of the impulse loading on the hull observed in the four blade analysis. Although it did not result in the complete elimination of potential flow-induced vibration sources, it was a step in the right direction.

In closing, the technology needed for accurate simulation of complex hydrodynamics through CFD software has come of age. The computational power and software is affordable for even small consulting firms. CFD can be used to give real-time recommendations to vessel owners and designers facing difficult flow related issues.

Major changes in a vessel design, such as the propeller, can be easily simulated with the results available in a few days. This has already shown itself to be a big cost saver to the owner and/or shipyard since it is cheaper to run software than to replace equipment in a trial and error process. In this case, if we had run the analysis before the client purchased the problematic four blade propeller, the cost savings would have been realised in the purchase of only the five blade propellers, and the elimination of the associated down time required to dry-dock the vessel for propeller replacement.

In addition to this exercise, ACMA has had very good experiences in utilising the software as a virtual tow tank, in which the impact of physical vessel modifications, such as hull form, and operational changes, such as trim, can be quickly modelled to estimate impact on powering requirements and fuel efficiency. ACMA has found a good correlation between the CFD-predicted values and real-world performance through direct in-house comparisons. CFD also allows for full scale simulation, eliminating issues of scaling that can be difficult to capture and handle in traditional model testing. ACMA expects CFD analysis to continue to expand its role as a key engineering tool in the future.

Authors' Biographies

Scott McClure, P.E., President of Alan C. McClure Associates (ACMA), has 29 years of experience in the offshore and marine industry. Donald Burris, P.E., Chief Hydrodynamicist at ACMA, has 10 years of experience in the offshore and marine industry. *NA*