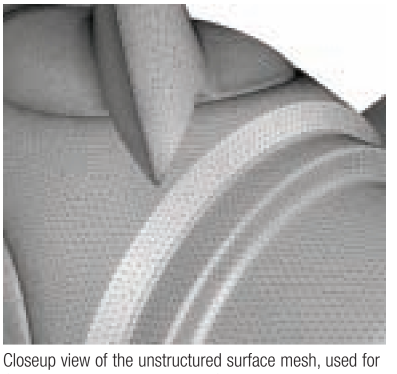
Propelling a More Efficient Fleet

Rolls-Royce uses simulation for propeller design to reduce marine fuel consumption.

By Johan Lundberg, CFD Engineer, and Per Aren, Project Manager in Hydrodynamic Design, Rolls-Royce Marine, Kristinehamn, Sweden

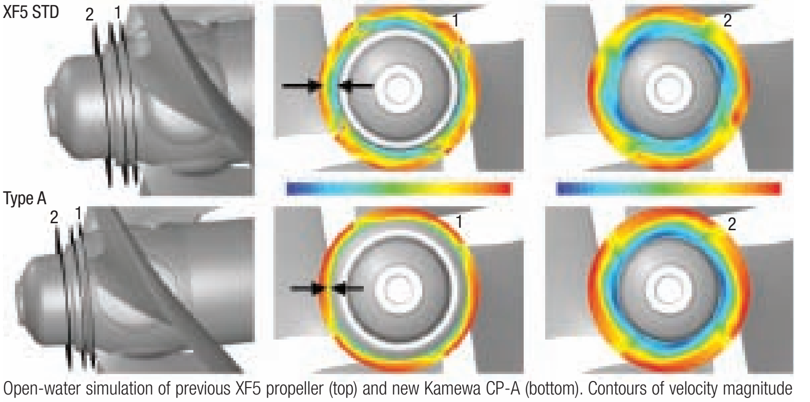
Rolls-Royce is a name associated worldwide with quality — and not limited just to automobiles. Through subsidiary Rolls-Royce Marine, its equipment is installed on 20,000 commercial and naval vessels around the world, and its comprehensive range of products includes gas turbine and diesel engines, nuclear propulsion systems, steering gears, stabilizers, thrusters, water jets, winches, cranes, rudders and main propeller systems. The company is a key part of the Rolls-Royce Group, with 7,000 employees serving 2,000 customers. The Rolls-Royce Hydrodynamic Research Center in Sweden is Rolls-Royce Marine's center of excellence in hydrodynamic propeller and waterjet design research. The center has combined the best of computational fluid dynamics (CFD) simulation and physical experiments to help the company develop the Kamewa CP-A, its latest controllable-pitch propeller (CPP).

A CPP is a special type of propeller with blades that can be rotated around their long axis to change their pitch. Changing the pitch makes it possible to provide high levels of efficiency and maneuverability for any speed and load condition. Stopping distance can be cut in half compared with a conventional fixed-pitch propeller. Traditionally, propeller development has been driven by a combination of physical experiments and potential flow analyses. Physical experiments have the advantage of being grounded in physical reality but involve the expense and time of building and testing a prototype. In addition, potential flow analysis is restricted in that it does not account for the full geometry of the propeller. CFD, on the other hand, can incorporate the full geometrical complexity of a propeller's operation. CFD can also provide much more detailed results than either physical experiments or potential flow analysis, such as flow velocities and pressure at every point in the problem domain, as well as the inclusion of viscous effects. The challenge at Rolls-Royce Marine was to incorporate the full complexity of flow around the propeller into the CFD model in order to accurately match physical experiments.

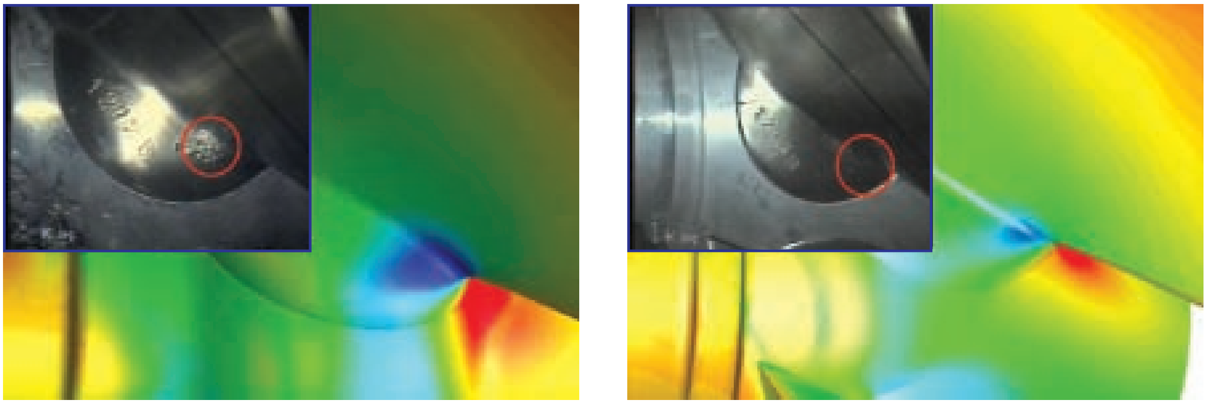


Closeup view of the unstructured surface mesh, used for CFD simulation of the Kamewa CP-A propeller

Engineers developed the CFD models using a hexcore volume mesh generated using TGrid pre-processing software from ANSYS. The hexcore mesh, which maintains cell surfaces perpendicular to the main flow in the core fluid region, kept the number of cells to a reasonable level. Rolls-Royce Marine used the full multi-grid initialization method together with the pressure-based coupled solver in ANSYS FLUENT software, which has proven to be both robust and fast for many applications. The engineering team simulated turbulence using the renormalization group (RNG) k-ε model,



Open-water simulation of previous XF5 propeller (top) and new Kamewa CP-A (bottom). Contours of velocity magnitude are shown at planes in two locations (1 and 2) for each design and indicate the low momentum, or viscous losses, close to the hub. The arrows indicate the thicknesses of the boundary layers.



Contours of pressure coefficient for the XF5 (left) and the new Kamewa CP-A (right). Insets: Photographs of the blade indicating the locations of the simulation where cavitation can be indicated (noticeable as pitting). ANSYS FLUENT results helped reduce pressure at the blade root in the CP-A design, as indicated by the lack of cavitation erosion present in the CP-A photo.

since it was considered to be stable and, thus, a conservative approach.

The engineering team began by analyzing propeller calculations for open-water operating conditions. These calculations considered the operation of the propeller in a uniform flow field without looking at the influence of the ship's hull. They then used the rotating reference frame method to simulate the rotating propeller. With this method, the team solved the flow equations in the rotating frame of the propeller blade. Integration of the calculated pressures and shear stresses on the propeller blades yielded thrust and torque, and the propeller's efficiency was then calculated using these values.

Design development then moved into a detailed study of the interaction between the propeller and ship appendages. The Rolls-Royce Marine team simulated the complete ship hull in order to calculate the effect of the wake field on the propeller design. Engineers used a sliding mesh model to simulate the operation of the propeller in the flow field under the influence of the ship hull. The sliding mesh model is a transient approach that calculates the flow field as one grid region rotates (or translates) relative to another. Historically, Rolls-Royce Marine designed the propeller as the last step in designing the ship, so there was no opportunity to improve the hull design to optimize the propulsion system. The ability to simulate the interaction of the propeller and hull has now made it possible to address such a concern.

The new Kamewa CP-A propeller from Rolls-Royce Marine

CFD simulations allowed Rolls-Royce Marine to evaluate a wide range of alternative hub geometries. The simulations also helped the engineering team reach a higher level of knowledge by providing far more information than physical tests could. For example, CFD made it feasible to easily determine the boundary layer in any prospective design. Generally, as the boundary layer gets thinner, the design becomes more efficient. By performing simulations of a number of different designs quickly, the team concluded that they could reduce the boundary layer and improve efficiency by modifying the hub contour.

Rolls-Royce Marine engineers were next concerned about the possibility of cavitation on the propeller hub caused by the boat's wake. Cavitation is the formation of vapor cavities in a liquid due to a localized reduction in fluid pressure below certain critical values. The vapor cavities collapse violently as they move to regions of higher pressure and generate pressure waves that cause localized stress on and damage to nearby components. ANSYS FLUENT results identified low-pressure areas in which cavitation could occur on the Kamewa CP-A hub. Changing the geometry increased the pressure above the critical level and eliminated cavitation, which made it possible to increase the load at the blade root to further improve efficiency.

According to a 2003 study from the University of Delaware, international commercial and military shipping fleets consume approximately 289 million metric tons of petroleum per year, which is more than twice the consumption of the entire population of Germany[1]. The ANSYS FLUENT simulations run on the modified propeller geometry predicted that the efficiency would increase by 1 percent to 1.5 percent, and physical experiments confirmed that this was, in fact, the case. This seemingly small improvement, however, has the potential to reduce fuel costs by several billion dollars if applied across the board to the world's commercial shipping fleets. It also has the opportunity to significantly reduce energy consumption and emissions of greenhouse gases. ■

References

[1] Corbett, J.J. and Koehler, H.W., "Updated Emissions from Ocean Shipping," *Journal of Geophysical Research – Atmospheres*, 108(D20), pp. 4650-4666, 2003.