Image ©Camera 4/Thonfeld.

Going for the **Gold**

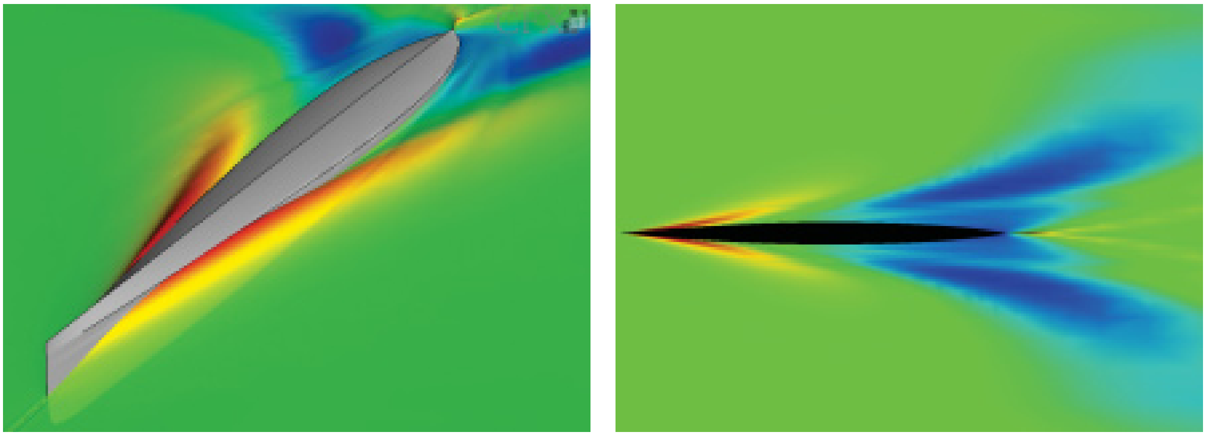
Simulation helps design low-drag canoes for Olympic-medal performance.

By Nicolas Warzecha
Institute for Research and Development of Sports Equipment, Berlin, Germany
Andreas Spille-Kohoff
CFX Berlin Software GmbH
Berlin, Germany

Competition among world-class athletes at the Olympics has become so intense that tiny variations in performance mark the difference between the gold medal winner and the also-rans. Relying heavily on computer simulation to reduce the air resistance of their bobsleds, the German national team leveraged their win in that sport to emerge triumphant in the 2006 Winter Olympics at Turin, Italy. Germany edged out the United States in overall gold medals, 11 to 9, and in total medals, 29 to 25.

The work in optimizing the performance of bobsleds was carried out by engineers at the Institute for Research and Development of Sports Equipment (known by its German acronym of FES) in Berlin, one of the world's leading centers for the development of sports equipment. Today, FES engineers are hard at work designing skiffs, canoes and sailboats that they hope will help produce a similar triumph at the 2008 Summer Olympics in Beijing, China.

To gain an edge in the canoe competition, FES engineers are using ANSYS CFX fluid simulation software to simulate the performance of various canoe designs. They selected this technology largely because it provides the powerful CFX Expression Language (CEL), which allows users to create their own physical models quickly from within the user interface, to add new variables, and to

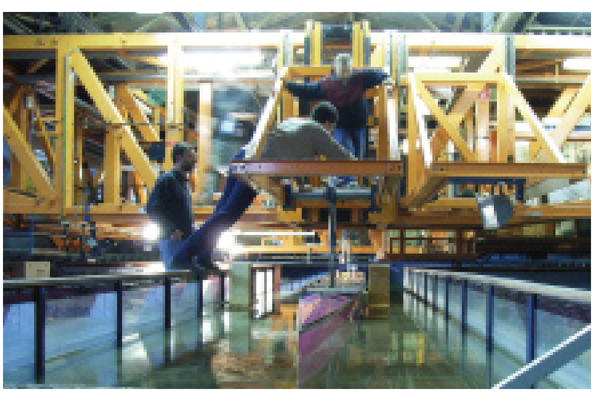


CFD results depict the wave pattern as it develops along a canoe body. Wave height is indicated by color, with blue denoting lowest areas and red indicating the highest areas.

define property relationships and boundary condition profiles. CEL goes beyond similar languages by allowing FORTRAN™ routines to be called, allowing other FORTRAN applications to be coupled to ANSYS CFX software.

The engineering team at FES began by simulating experiments involving towing a canoe, which eliminated the challenge of simulating the effect of the paddlers' strokes on the boat's motion. They used ANSYS ICEM CFD Hexa software to create a block-structured grid model with 3 million elements and used the free-surface multiphase model of the ANSYS CFX product to analyze both the motion of the water and the air trapped by the movement of the boat and the water. This simulation showed a very good correlation with the drag measured in the towing experiments. All cases were simulated in parallel on a 64-bit Linux® cluster, whose installation was supported by CFX Berlin. The best results were acquired by running the CFX solver on 10 to 20 mesh-partitions, depending on the size of the grid. Using this approach, a transient simulation representing a 10-second real-time interval for a moving boat could be performed in one to two days.

The wetted surface of the canoe in this model did not match the experiments, a conclusion that was expected since this simplified model did not account for the forces and moments acting on the boat. So FES engineers simulated the boat at its position with the ANSYS CFX solver to calculate these forces and moments and estimated the new position of the boat — that is, sink and trim values. They continued with a series of manual steps that slowly converged to a final position showing good agreement with experiments. Once they had determined that this approach provided realistic results, FES automated the analysis, with the assistance of CFX Berlin, by writing CEL expressions and some pieces of FORTRAN code that performed all of these steps in the same way but much faster and automatically within the ANSYS CFX solver. With this approach, it was possible to evaluate and compare the performance of several alternative designs.



Experimental testing area used to study water flow and wave formation around a boat as well as total drag

The analysis performed by FES provides the drag as well as complete information on the movement of the water around the boat, the position of the boat and the forces acting on the boat. In particular, computational fluid dynamics (CFD) makes it possible to measure the bow wave, aft wave and wake of the canoe to a high level of precision. Simulation provides engineers with good indications of what is causing drag in a particular design and what aspects of the design they should change to improve it.

In the past, designing these boats was based largely on trial-and-error prototyping. Using simulation, FES engineers were able to quickly design a new skiff that ANSYS CFX software predicts will provide a 3 percent improvement in drag. A prototype of this boat is currently under construction. After completion of the prototype, physical testing will be used to verify the simulation results. In the meantime, FES engineers are working to expand the scope of CFD analysis to analyze other effects that are difficult or impossible to measure, such as the effect of initiating paddling forces at different times and variations in the velocity of the boat. Including these effects in the analysis may make it possible to advance the performance of canoes and athletes to even higher levels. ■