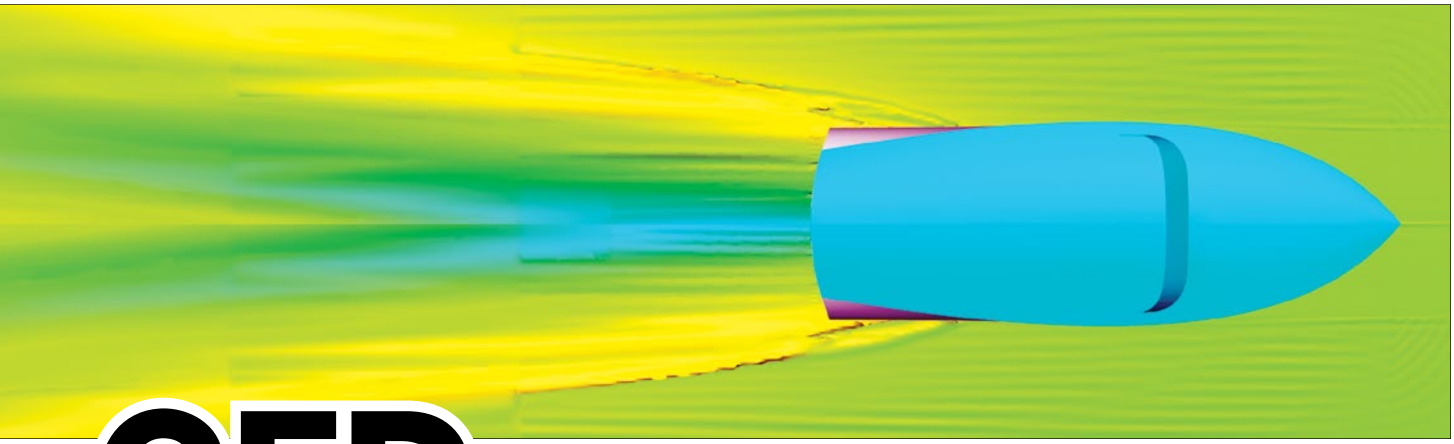


CFD in the Small Design Office

The expense and training that computational fluid dynamics software requires is now justifiable for even a one-man design office.

Text and graphics by Steve Killing (except where noted)

It's hard to believe, but when I started designing boats at C&C Yachts in 1973, I used a Teflon-coated aluminum slide rule for hydrostatics calculations, and that was pretty high-tech. Just determining how a new design would float was a full day's task. In the successive 48 years, new tools have grown in accuracy and usefulness as we progressed through Texas Instruments calculators, Hewlett-Packard programmable calculators, a Radio Shack TRS-80 desktop computer, and countless iterations of hardware and software until today when I am running Rhino 3D and Orca3D Marine CFD on a 16-core gaming computer.

At each step along the way, new tools have helped designers create and analyze efficient, useful boats. Through the 1970s, tank-testing was the preferred method to predict the performance of new designs, although the cost and time required meant that it was practical only for large projects. As scale effect problems were discovered (notably by Karl Kirkman at Hydronautics), there was a move to larger models, typically one-third scale, and designers followed suit when budgets permitted. At the time, there was talk of computer analysis, but it was a long way off.

Through many America's Cup campaigns in the 1980s and '90s, tank-testing held its own, but computational fluid dynamics (CFD) began to improve. More and more, the results showed that the correlation between digital modeling and both tank-testing and real on-the-water performance was quite good. It took time for designers to become comfortable enough with CFD to rely on the results. My own involvement with the New Zealand America's Cup challenge in 2013 (the San Francisco event) exposed me to some heavy use of CFD by enthusiasts in the field. At that point, the science of CFD was best left to the experts, and the whole thing was a bit of a mystery. Looking over the shoulder of the operator, one witnessed an array of decisions on grid patterns, step size, and more numbers than images. For those of us who need a visual reference, it was rather a blur.

I admit that 20 years ago I was a skeptic. I had yet to see computer predictions that were solidly borne out on the water, and a combination of the cumbersome interface and the cost put it out of reach for me as an independent designer. Yet today I am running high-powered CFD in my one-man office—and loving it. What changed?

The software that does the work is

still a sophisticated Reynolds-averaged Navier-Stokes solver developed by Simerics Inc., but the difference is an interface made by Orca3D that launches right out of my Rhino3 design software. I can use the tool with confidence to assess candidate designs or design modifications.

If you had asked me even five years ago whether I thought I would be running CFD in my office, the answer would have been a definite no. I didn't have the knowledge, the cash, or the computer power to make it viable. But a lot has happened in a short time to make CFD practical for the small design office. Of course, the same is true for large offices—the software I rely on is also used by major recreational boatbuilders, shipyards, the USCG, and university naval architecture programs.

Although you don't have to be a CFD expert to run the software, it is important that you have enough knowledge to avoid accidental misinformed conclusions—and that knowledge comes from the software vendor. Any computer program that is somewhat advanced needs to have good support from the creators or tech-support team. There is no point in making great color images if you don't have confidence in the results. In my case, the software all came from Orca3D, and even though it seemed like Bruce Hays and Larry Leibman (the principal naval architects at Orca3D) were just teaching me how to use the software, I realized they were also teaching me the ins and outs of CFD more broadly. In this case, both Simerics and Orca3D have been more supportive than I would have expected, providing valuable orientation sessions but also problem-solving and suggestions for my specific applications. And when I did something stupid, good humor and understanding prevailed.

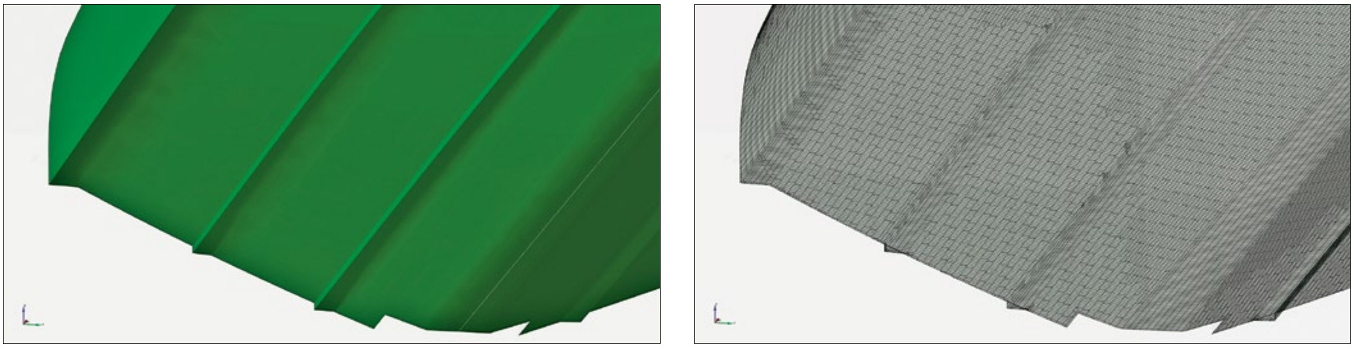
The first step with any new software is overcoming the challenges of a new computer interface. Just maneuvering around the screen will, no doubt, be controlled by some digital derivative of Murphy's Law that says whatever way you zoom in and out in your current software, it will be the opposite on the one you are learning.

For me, the time from starting to use CFD to having confidence in results of my work was about two months. Most of that was spent learning from the vendor the important steps and measures to ensure reliable results.

It turns out that gridding the hull and appendages (dividing into appropriate cells for analysis) is a compromise between analysis time and accuracy. The good news is that the process is automatic within general categories like coarse, normal, fine, and extra fine. Time step size is another key factor in determining how quickly the results converge or whether they converge at all. Conveniently, the software automates the selection of an appropriate time step as well.

The issue of cost came up early in my experimentation with CFD. In short, this is the most expensive software in my office, and on top of that, I don't even own it—I lease it three months or a year at a time. For the record, I am the kind of designer who likes to pay for something once and then use it when I want. So, for me to be comfortable leasing software, it must be something I see as very valuable. But as a direct cost comparison, one day of tank-testing will cost you more than a three-month CFD license. That seemed like a good trade-off to me. But even then, to make the initial financial commitment to the software, the support of one manufacturer who wanted to advance the analysis of their boats, made all the difference. They committed to enough work that it was easy to make the leap. The big surprise was the number of investigations for other clients that I now had the correct tool to tackle. It meant that I quickly found more work for the CFD software, and now, after four months of use, I have invested in a separate computer that runs CFD almost 24 hours a day.

One of the strengths of CFD is that it isn't limited to just one type of boat; because it models first-principles physics, the same software can be used to analyze a sailboat, a superyacht, a stepped hull, a pontoon boat, etc.



Facing page—*The wake, whether behind a planing powerboat or a displacement hull, is accurately modeled in CFD software.* ***Left***—*The CAD surface from Rhino3D is of the aft section of a powerboat with strakes and, on the* ***right,*** *the grid produced by Orca3D Marine CFD. Typical grid size is about three million cells.*

The following are a few examples of projects I have used CFD for so far.

The Rowing Shell

The vessels I design in this office span a wide range of propulsive power, from high-speed powerboats to human-powered craft that seldom see more than half a horsepower. Even in an Olympic rowing shell there is so little power available (compared with an electric or combustion engine) that every detail of hull shape becomes important. It was these sleek carbon shells I was designing for Canadian builder Hudson (London, Ontario) that pushed me to take the final leap into the world of CFD. I had been designing shells for them for 10 years and was seriously looking at bringing CFD analysis into my own office. The technology and its influence on the design process were not new to them—the builder had used dynamic CFD on its new shells in the past but primarily for confirmation of design decisions already made. Applied in that way, it lacked some usefulness because of difficulties of operation and the speed of execution.

When we revisited the possibility of applying CFD, I first tried going back to a simpler solution—a software analysis tool based on Michell's integral for predicting wave drag on slender vessels. The results were acceptable and permitted us to make performance improvements, but the limitations were glaring—there was no dynamic consideration or detailed shape assessment.

We decided to look more closely at the new Orca3D Marine CFD. With its unintimidating interface and high-speed operation, it meant that CFD was now a practical design tool and a good fit with my desire to be directly involved with the analysis.

In 2014, we had been fortunate to be able to tank-test two different full-scale rowing shell designs at the National Research Council (NRC) towing tank in St. John's, Newfoundland, Canada. These results provided a clear benchmark we could compare with and validate our Michell's code. Now we could also use it against the CFD results we were getting from the new software (see **Figure 1**).

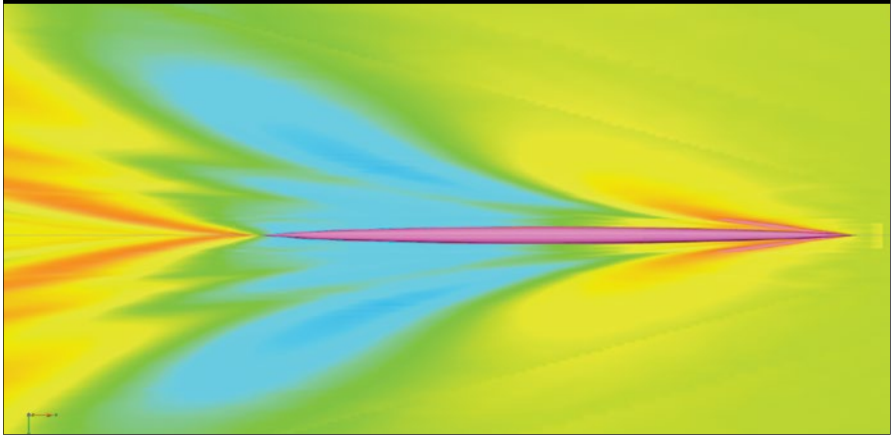

COURTESY USROWING

Rowing shells, sailboats, and powerboats all float in the same water, and all can benefit from CFD modeling as they are designed.


COURTESY NATIONAL RESEARCH COUNCIL, CANADA

In this full-size rowing shell in the towing tank, midtest, note the minimal wave production on the narrow hull that correlates well with the CFD run.

Figure 1. Rowing Shell Wake



Most of the resistance of the slender rowing shell is controlled by the frictional drag of water next to the hull. Even though this wave pattern looks extreme, it accounts for only about 15% of the drag. Compare this wave train with the powerboat wake, and you can see planing is quite a different phenomenon from the movement of a displacement hull.

First, some context: The margin of improvement likely in a new rowing shell design is on the order of 1% or less. It seems small, but a 1% change in speed over a 2,000m (1.24-mile) course results in a victory of 20m (65.62′), which any athlete would love to experience. So, I am excited if I can make a 0.5% speed improvement. That means I want whatever speed and drag analysis tool we use to be consistent to about one-tenth of one percent if we are to have confidence in the results. I am less concerned that the absolute value of the answer is exactly correct, but when I compare two candidate designs, the relative values need to be dependable.

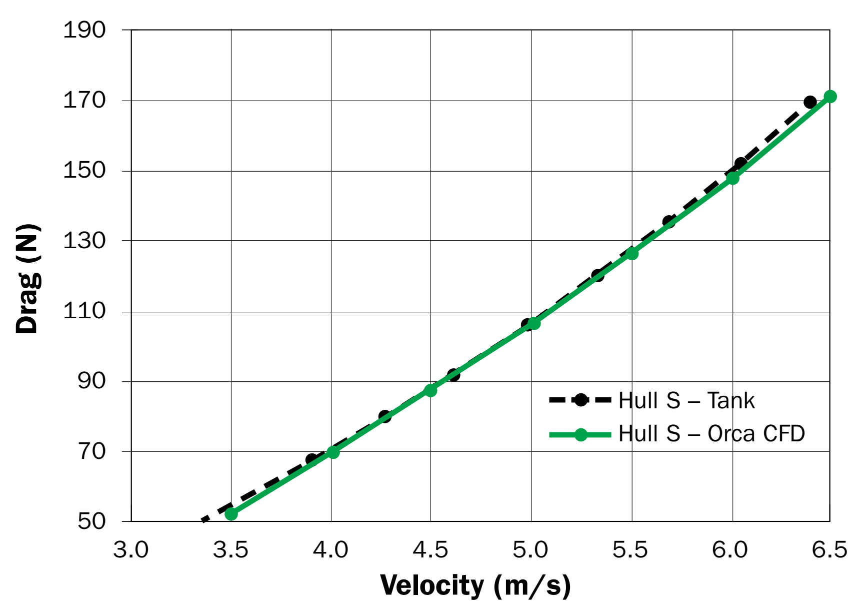
In the plot of Drag versus Speed (for Hull S) in **Figure 2**, you can see the comparison of the steady state flat-water free-to-trim-and-heave tests for the tank versus the CFD runs with excellent correlation. On my 16-core computer, runs take significant time when compared to parametric methods but are relatively fast for CFD. Each point on the graph took eight hours to converge to 0.1% variation. The time for a run to complete is inversely proportional to the number of cores in the computer, because the software can spread the calculations out to each core so they run in parallel. While it's possible to run the software on a four-core laptop, the cost of 16-core or even 64-core workstations is now within reach of even small design offices.

One of the intriguing challenges in assessing the performance of a rowing shell over a 2,000m Olympic course is the dynamic motion of the boat. If you have rowed any kind of boat, you know that the propulsion force from the oars is not steady. In fact, on the return stroke it is zero or negative in value. This periodic force from the oars, combined with the rower's momentum change, causes the boat to pitch a significant amount (as much as 2″/50mm) at the bow. Various towing-tank techniques have tried to simulate this motion, but as far as I know, none have succeeded, and certainly the time and expense of such testing are prohibitive.

These factors presented some significant challenges for CFD as well, but having the technology close at hand in the office meant we could afford lots of experimentation as we looked for solutions.

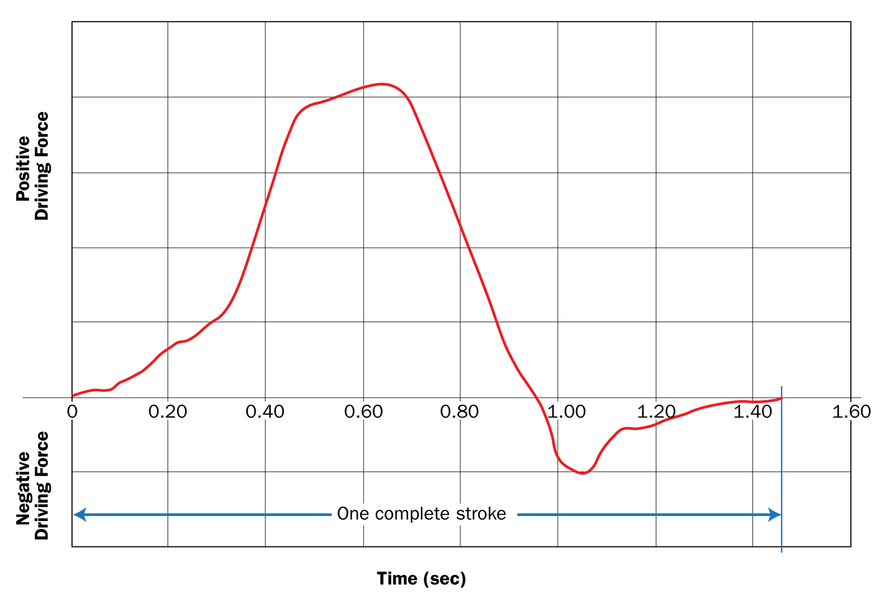
Once we had confidence in the constant velocity CFD runs versus the tank-testing, I moved on to the dynamic runs. The builder, Hudson, has always embraced the technical side of boat design and building, so CFD fit well with their priorities and past practices. They have outfitted many

Figure 2. Drag versus Speed – Hull S



This plot of drag versus speed for a rowing shell shows good correlation between the towing-tank results and the CFD analysis. Normal competition speed for this hull is about 5.2 m/sec. It is not clear at the higher speeds whether the tank or CFD provides the more accurate solution.

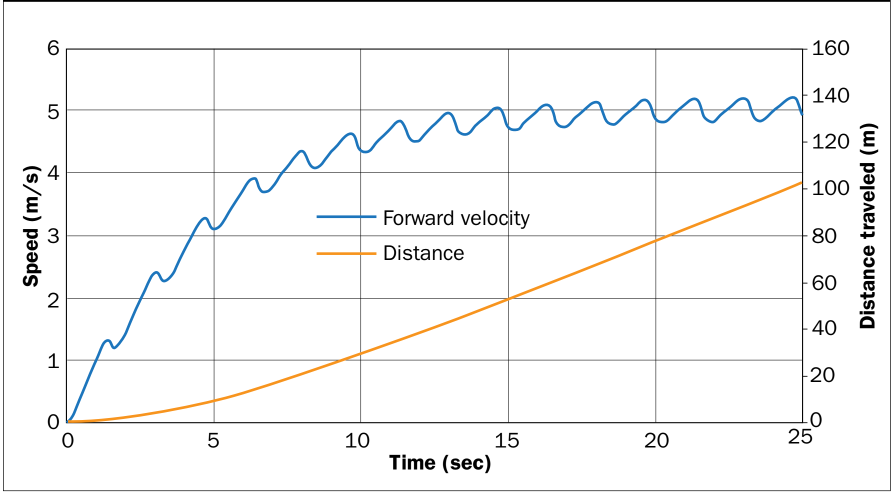
Figure 3. Rowing Oarlock Force



In this typical driving force for a single rowing shell, at time zero, the blade enters the water, and peak force is at 0.65 second. The time from 0.95 to 1.45 seconds corresponds to the return stroke with the blade out of the water exerting a negative force due to the wind force and momentum of the oar. This variable force propelling the rowing shell is repeated over and over (in this case, every 1.45 seconds).

shells with instrumentation to record the loads exerted by experienced athletes. To correctly model the forces exerted on a shell, this requires force gauges on the oarlocks, the footstretchers, and the seat. We use the Peach system, developed in the United Kingdom, which can be installed in a morning to record the loads. At this writing, we have just finished building an in-house instrumented seat to record the vertical loads on the sliding seat—the one measurement missing in our matrix. Just to be clear, this is a one-time experiment to record the force and moment produced by a rower so the data can be used as input to the CFD model. With some careful

Figure 4. Single Rowing Shell – Speed versus Time



The periodic rowing stroke results in a variable but increasing speed. Starting from rest, the boat accelerates over 15 strokes to full speed in this CFD run. The orange curve is the distance traveled (in this case, terminating at 102.49m) and can be used as a performance measure to compare hulls of various shapes.

analysis, this provides the net cyclical force in the direction of motion and the moment that contributes to the pitching motion. You can witness this vertical bow movement by watching videos of any Olympic rowing race.

In **Figures 3** and **4** our CFD runs with a representative variable oar force illustrate the dynamic motion during the rowing stroke as the boat accelerates and decelerates with each stroke.

To assess new designs, one could use the last three strokes in the plot of the Single Rowing Shell – Speed versus Time (Figure 4) and average the speed results. But we found that averaging is very sensitive to the segment of data selected when trying to find small improvements. We have discovered that the clearest method of performance analysis is to apply the forces and moments to the CFD model and let the model progress through the cyclical speed variations for exactly 15 strokes. The distance traveled is the performance measure.

The Planing Powerboat

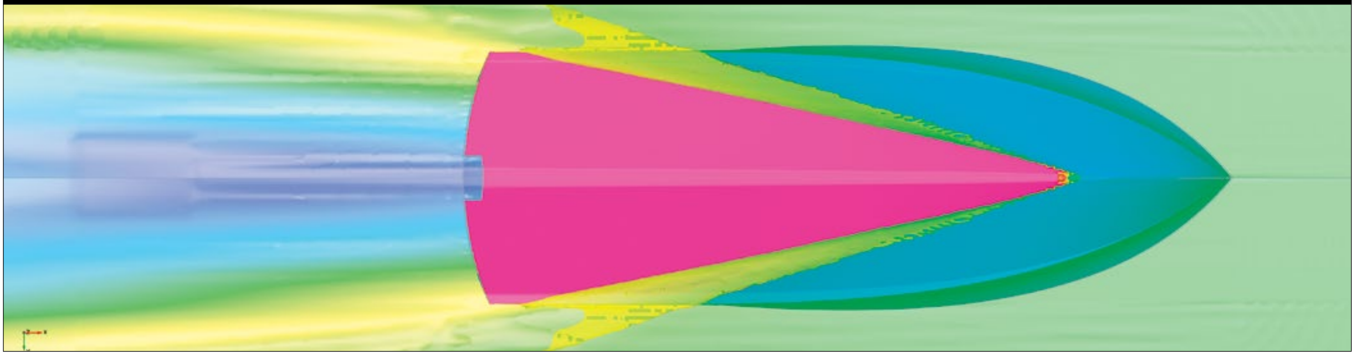

ROBERT NELSON, COURTESY CLARION BOATS

The Clarion 20 (6.1m), boat number one, is shown in her cottage-country element.

The inland lakes of Muskoka, Ontario (I live just south of them), are a perfect venue for the gentleman's runabouts that were popular in the 1920s through the 1950s. Although many boats have been restored, we are running short of relics behind barns and so are designing new boats with the flavor of the past. This 20-footer (6.1m) for Clarion Boats (Muskoka, Ontario) is a departure from what I normally design for them, in that we incorporated both a sterndrive for ease of maneuvering and lifting strakes for a drier, faster ride (on the traditional hull). In the design phase, I was getting some pushback from the builder as to whether we really needed the lifting strakes, which would significantly complicate building the hull. I wanted some CFD confirmation of the benefit. After running several combinations of strakes through the program, the visuals of the running surface clearly illustrated the value in removing solid water and spray from the bottom. With both boats at a constant speed of 30 knots, the boat with strakes showed a 13% reduction in drag.

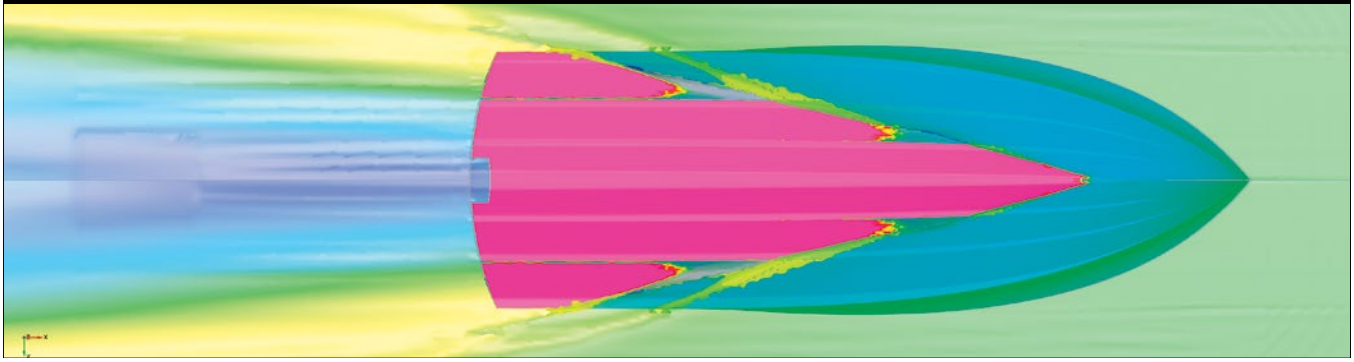
At a boat speed of 30 knots, the heave (vertical rise of the boat) for both boats is 7.4″ (18.8cm) from the at-rest condition. The bow-up attitude is also almost identical at 1° for the bare hull and 1.05° for the straked hull. But the big change comes from the reduction in wetted area and the associated drag, as shown by the resistance of 616 lbs (279.41 kg) for the bare hull and 536 lbs (243.13 kg) for the straked hull (**Figures 5** and **6**). The additional benefit is the drier ride with the spray deflected downward.

Figure 5. Without Strakes



CFD output of the boat running without strakes has 616 lbs (279.41 kg) of drag. The magenta area is wet, and the blue area is dry, while the yellow area is some level of spray.

Figure 6. With Strakes



CFD output of the boat running with strakes has 536 lbs (243.13 kg) of drag, and the reduction in wetted area is apparent along with reduced spray. Note that there is some air on the outside of the outboard strakes that makes its way along the strakes to the transom. If twin engines were installed, that could result in air being drawn into the propellers.

ROBERT NELSON, COURTESY CLARION BOATS

A dry ride and a happy client are nice rewards for some careful analysis.

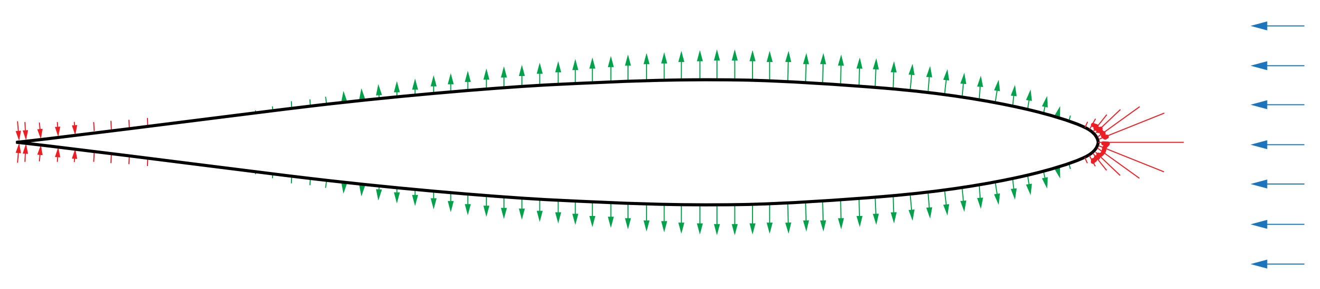
These drag results will not be a surprise to powerboat designers, but the benefit for the client is being able to simultaneously quantify and visualize the results when the designer suggests changing a hull shape that has worked in the past. There is often a great reluctance to alter something that is working. And yes, I could have gone to a CFD specialist to analyze the problem. But as a designer, having the tool in your own office, watching it iterate through the solution, seeing an animation of the bow-up attitude on take-off, gives you time and context to think of new solutions and options. And if you have a new idea, you can just shift over to the designated CFD computer and try it out. Typical run time is six hours per speed for a towed run and 10 hours for a powered run on a planing powerboat (simulating being pushed by a propeller).

Sailboat Keel Detail

Most sailboat keels, including those I design, have what I would call a soft toe. That is, the lower forward corner of the keel has been rounded and faired to ease the flow transition as it meets the bottom of the keel. Prior to my use of CFD, I had no way to assess the benefit of the shaping, if indeed there is one. This question came up in a recent project. It led to an investigation I could readily execute using the CFD capacity in my office.

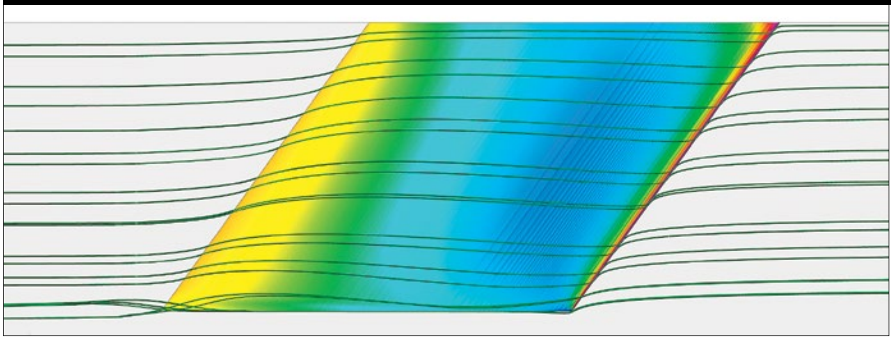
To start, I designed two versions of the keel, one with a sharp toe and the other with a soft toe. CFD runs were made at zero leeway representing a downwind run and at 2° leeway for an efficient upwind course. But before we look at the CFD, study **Figure 7**, which gives some context to the changes in pressure drag the software reveals.

Figure 7. Pressures on a Keel Foil



To help explain the change in pressure drag we see in CFD, this cross section, from foil analysis software Profili, shows red lines as pressure toward the surface and the green away from the surface. The red pressure at the (right-hand) leading edge is speed-reducing drag. The green pressure forward of the maximum thickness is producing a thrust, while the green pressure aft of the maximum thickness is contributing to drag.

Figure 8. Sharp Toe Downwind



The streamlines indicate some nonintuitive flow direction for the keel at zero degrees of yaw.

Streamlines shown in **Figures 8** and **9** are generated at random heights, so the spacing of them is not significant, but the direction and interaction are indicative. The Sharp Toe Downwind (Figure 8) streamlines at the aft end show some drag-inducing interaction as the flow comes off the bottom of the keel. Compare that with the smoother shape of the Soft Toe Downwind (Figure 9) streamlines.

In the images of the two keels, red and orange are positive pressure toward the keel's surface, green is neutral, and blues are negative pressure (away from the surface). The soft toe in particular exhibits less detrimental red pressure in the area of the toe. Total drags are 39.36 N for the sharp toe and 37.74 N for the soft toe.

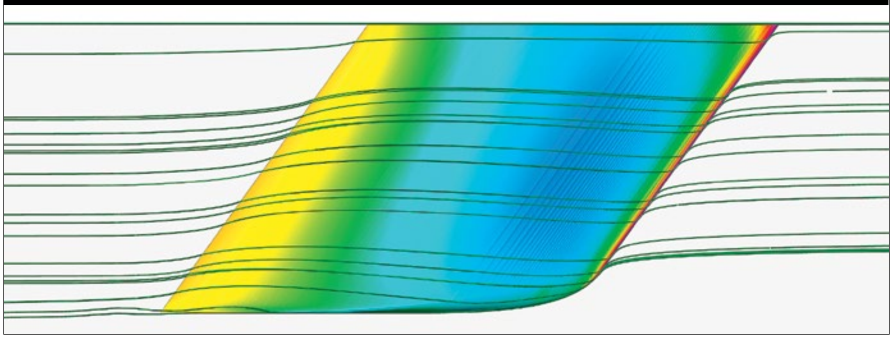
Analysis was also done for 2° leeway with both lift and drag measured. The soft toe keel showed less drag and less lift (presumably because of the reduced area). The important measure upwind is the lift-to-drag ratio, and the soft toe keel wins out at 7.22 versus 6.96 for the sharp toe.

With this CFD numerical confirmation of a keel tip detail that most designers already feel is correct, the incentive is set to try further investigations into more adventurous tip details with the goal of further reducing drag.

◆

Based on my recent experience, I can affirm that it's now practical and affordable, even for a small design office, to bring CFD inhouse. For

Figure 9. Soft Toe Downwind



Streamlines at the aft lower point of the keel are smoother than for the sharp toe. The dynamic pressure, indicated by the colored zones, is more advantageous for this toe treatment. The 4% reduction in drag over the sharp toe is entirely from the reduced pressure drag.

numerical confirmation of assumptions that designers have been making, and to analyze innovative shapes and features that cannot be readily assessed using traditional empirical methods, there is no better tool. This allows designers to answer not just whether something is better but how much better. While these tools can save time and money previously spent on model or full-scale testing and modifications, perhaps the most exciting benefit of having CFD right in your design office is the opportunity to learn, and maybe even stumble upon, new things that are not intuitively obvious.

About the Author: *Steve Killing is a yacht designer living in Ontario, Canada. He keeps his one-man design office busy by working on a variety of marine projects: cruising and racing sailboats including the America's Cup and C-Class catamarans, classic mahogany runabouts, modern electric boats, canoes, kayaks, and rowing shells.*

Resources

clarionboats.com
hudsonboatworks.com
orca3d.com
profili2.com
stevekilling.com