THE ROYAL INSTITUTION OF NAVAL ARCHITECTS

6 DEGREE OF FREEDOM CFD APPLIED TO THE DESIGN OF AN IMOCA OPEN 60

R Azcueta and **R Schutt,** Cape Horn Engineering, Spain

SUMMARY

This paper presents the application of multi-degree of freedom Computational Fluid Dynamics (CFD) simulations to the design of high performance sailing yachts. It aims to demonstrate that simulations are a preferred alternative to tank testing when assessing overall boat performance. The methodology is based on experience gathered over the last 10 years of designing high end racing boats, especially in America's Cup and Volvo Ocean Race campaigns. When looking to maximize performance yacht designers can benefit from recent advances in CFD. Cape Horn Engineering is a company that specializes in hydrodynamic and aerodynamic CFD for the marine industry. A general yacht design philosophy based on simulations is presented along with specific applications of 5 and 6 degree of freedom (DoF) simulations in the design of an IMOCA Open 60.

1. INTRODUCTION

The effect of Reynolds-Averaged Navier-Stokes (RANS) free surface flow simulations on the design of high performance racing boats is undeniable. Especially in the America's Cup and Volvo Ocean Race, the application of these CFD techniques has led to unparalleled success.
The results achieved in these campaigns have made engineers and yacht designers extremely confident that simulations, when applied diligently, can be a superior alternative to traditional tank and wind tunnel testing.
Simulation development in the America's Cup environment requires rigorous testing and validation to keep teams competitive at the highest level. The difference between two design candidates can be smaller than the experimental error when running the same towing tank test twice. Numerical simulations eliminate this inconsistency and offer fundamental advantages over physical testing.
Recently, industry leading design projects, in which the authors were involved, relied solely on CFD simulations. At Cape Horn Engineering we have forgone towing tank and wind tunnel tests in favor of an exclusively CFD based design philosophy. Our simulations are cheaper, faster, and more reliable than traditional tests. Simulations are run at full scale, which eliminates the error inherent in scaled test results. Enhanced flow visualization and force decomposition gives designers a much greater understanding of flow phenomena. This relatively new technology can be successfully taken from the racing environment and used for any high performance sailing project.
In addition to maximizing velocity performance, CFD simulations can successfully be used to assess power requirements, balance, behavior in waves, structural loads, windage, and water on deck.

In this paper we will describe the underlying numerical methods first, then explain the general CFD based design philosophy used for a recent Open 60 racing yacht. Finally, examples of the use of new 5 and 6-DoF simulations in the design process will be given.

2. NUMERICAL METHOD

For our simulations we use the commercial RANS codes from CD-adapco, Comet and Star-ccm+.
We have used Comet for more that 10 years with great success, applications and validation are presented in several papers, see [1-4] for details.
Focus is now mainly directed towards the use of Star-ccm+, which builds on Comet's strengths while developing a more robust, user friendly package. Star-ccm+ includes the latest physical models and solver technology (turbulence models, transition models, cavitation, integrated unstructured volume meshing with both polyhedral and trimmed cell approaches, etc).
Both codes have shown to be very flexible and efficient to use. Functionality can be extended with Fortran and Java user programming.
Comet's cell-based numerical capability is still our tool of choice when small design changes need to be evaluated. It is highly accurate and fast with free surface computations. Star-ccm+ is better geared towards complex geometries and complex physics.
Numerical computations for marine applications involve the coupling of a RANS free surface flow solver to a body motion solver and a mesh motion/deformation solver. The RANS equations are solved using a finite volume approach. Both Star-ccm+ and Comet handle any cell type or topology, including polyhedral elements with any number of faces.
Accurate viscous solutions require refined mesh resolution near the wall. Turbulence models, K-Epsilon and K-Omega, along with wall functions, are generally

used to decrease computational time and maintain adequate accuracy. When extremely accurate boundary layer analysis is needed, transition is predicted with a Gamma ReTheta model coupled to the SST K-Omega turbulence model using a mesh with tens of millions of boundary layer cells.

Free surface simulations have long been the strong point of Comet. Comet and Star-ccm+ both use a similar interface capturing approach; the free surface is modeled using a volume of fluid (VoF) method. The air-water interface is kept sharp using an high-order advection scheme based on the high resolution interface capturing (HRIC) interpolation scheme, see [5] for more details.

For fluid body interaction problems, the equations of motion are solved using a body motion solver. Rigid body motion resulting from viscous and pressure forces acting on the body is calculated by integrating equations of linear and angular momentum. External forces and moments from sails or propellers are added during this step. Basic simulations are free to sink and trim (2-DoF), while more advanced seakeeping and maneuvering simulations can be completely free (6-DoF with moving control surfaces).

In calm water cases where dynamic results are not needed damping is added to the equations of motion to increase the convergence rate and decrease the total computational time. In calm water and in waves we use a single moving mesh strategy when solving for single body motion. The mesh is not deformed but moved. This is a robust and fast method to handle the motion of the boat. Large motion does not degrade mesh quality and there is no deformation overhead.

All of our simulations are processed in parallel on 4 to 16 cores; with a large in-house cluster we can processes many cases per day.

3. CFD BASED DESIGN

Cape Horn Engineering has been involved in Volvo Ocean Race and America's Cup design campaings in association with Juan Yacht Design since 2003. During the ABN Amro and BMW Oracle Racing campaigns a thorough validation with tank testing was conducted. In more recent campaigns with Ericsson Racing Team in the Volvo and Team Origin in the America's Cup the entire design process was based on CFD simulations.

The following is a description of our design philosophy in a recent IMOCA Open 60 project.

All hydrodynamic and aerodynamic simulations are run separately. Hydrodynamic simulations are used to research hull form, yacht behavior in waves, and appendage shape and position. Early in the design process aerodynamic simulations are used to determine the sail forces which are then used as input to hydrodynamic simulations. Later we return to aerodynamic simulations to optimize sail shapes and investigate new sail concepts.

Our hydrodynamic cases are run using full size hull models with rudders, keels, and foils. In aerodynamic simulations we model all geometry above the static water plane including the sails, mast, boom, deck, and hull. The rig is tilted to the correct attitude, accurately modeling heel, pitch, and yaw. A varying wind profile is used to account for the boundary layer along the surface of the water.

For sail analysis we use parametric modeling and fluid structure interaction (FSI) codes. In parametric models we use CFD to find the optimal aerodynamic sail shape. Sail designers then develop a sail with the appropriate structural elements in order to achieve this optimal 'flying' shape. Parametric variation is done either by using a predefined matrix of variations, or by incorporating an optimizing algorithm in an iterative loop. In FSI models, the CFD code passes pressure forces into a finite element model which calculates the deformed shape of the sail. The new deformed shape is trimmed by the sail designer and fed back into the CFD simulation. This cycle is repeated four to five times until convergence is reached.

In hull shape studies we use fully appended models with a reference set of rudders, keels, and foils. For appendage studies we use a set of reference hulls and change appendage concepts, shapes, positions, and orientations. Each variation is evaluated by comparing the resulting forces (drag, side force, roll, and yaw moments) and by comparing the flow characteristics using stream lines and other visual techniques.

The IMOCA Open 60 Class boats present a complex design problem. Compared to previous America's Cup Yachts there are many more design variables. Open 60's are designed to a box rule and experience sailing conditions from all over the world. Boat speed ranges from 6 knots as a displacement hull to over 30 knots planing and surfing down waves. The boats have canting keels and water ballast which drastically changes the displacement of the boat and the center of gravity. All of these variables lead to very large testing matrices. Here CFD becomes very attractive; each variable usually can be changed simply by changing a number and running the simulation again.

Our hull shape research program is quite extensive. The design spiral begins with the required transverse stability and waterline beam. We then obtain accurate sail force coefficients using our own aerodynamic simulations. Different sail sets are tested for upwind and downwind sailing in light and heavy air conditions. This is important, especially for very beamy boats, because the sail forces can have a large effect on longitudinal trim which in turn significantly affects the drag.

The hull shape investigation continues with studies of volume distribution, prismatic coefficient, transom width and immersion, bow fullness, etc. Hull shapes are organized with parent hull shapes and their derivatives. This allows for easy analysis of trends and performance drivers and the final selection of a hull shape. After the final hull has been chosen and the lines have been sent to the builder, research continues with appendages and sails. We investigate the size, shape, and position of the keel, bulb, rudders and dagger boards. We adjust the

transverse and longitudinal inclination , alignment of the keel cant axis, pitch of the bulb, angle of attack of the dagger boards, etc. Different solutions and details for the attachment of foils to the hull are investigated.
Candidate boats are then run through race simulations to determine which design is the best. A Velocity Prediction Program, or VPP, is used to analyze trade-offs, such as stability versus drag, and to determine optimum balance. A Router program simulates the best course for each hull using statistical weather data for relevant parts of the world and then compares the time needed to complete the course for each hull. Thus, the overall winner of the race is found in probabilistic terms. Other design variations, such as the position of a dagger board, are more straightforward to analyze. Usually it suffices to compare the amount of drag at a given side force to draw conclusions.
Finally, a set of seakeeping simulations are performed to investigate the dynamic behavior of the final candidate hulls. These simulations verify that what is good in calm water does not become detrimental in waves. Unlike a towing tank, no other simplifications are necessary; in our simulations, any wave direction is allowed, the boat is at full scale, the center of mass is in the right position and the moments of inertia will be the actual estimated values.
In many cases, sail and appendage designs and modifications are given a reality check when tested on the trial boat in real sailing conditions. During this on-the-water training period we are given valuable feedback from the real world that keeps us motivated and focused.

4. 6-DoF SIMULATION METHOD

In traditional towing tank tests boats are only free to move in heave and pitch. This results in test conditions that do not accurately represent real sailing conditions. With 6-DoF simulations the boat is free to move as it would in the ocean, all forces and moments are balanced. Boat designs can be very accurately compared by applying sail forces and letting the boats move naturally until they reach equilibrium at their steady-state velocity and attitude.

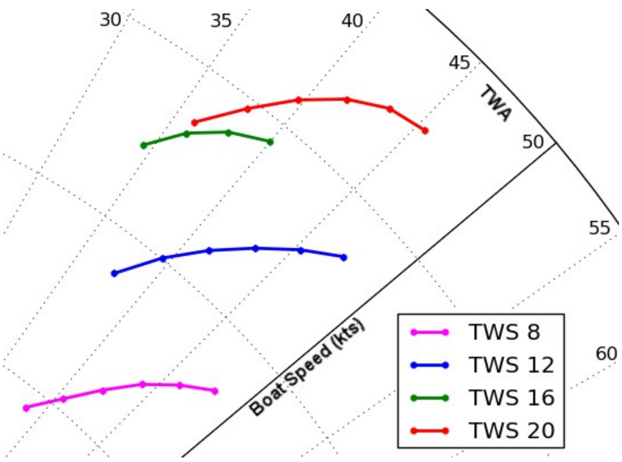

Figure 1: 6-DoF Polar

All effects of a design change are readily apparent, hydrodynamic results do not need to be post-processed by a VPP, which could hide subtle secondary effects.
A series of 6-DoF simulations can be used as a highly accurate replacement for a VPP. Figure 1 shows a polar plot generated from 18 separate simulations. A sweep of true wind angles at each wind speed is used to find the maximum VMG.

4.1 AERODYNAMIC FORCES

In 5 and 6-DoF cases the boat is propelled by sail forces interpolated from previous aero CFD simulations. An appropriate sail is chosen for each sailing condition; if the sail is not already in the library of sails results, a matrix of steady state cases is run across varying heel and apparent wind angles and added to the library. Our sail library currently contains hundreds of sails and is always growing.
While the 6-DoF hydro case runs, an 'aerobox' module uses the current orientation and velocity of the boat to calculate the forces and moments generated by the chosen sail. The aerobox takes force coefficients and centers of effort from the matrix of heel and apparent wind angles. The apparent wind speed is then used to calculate forces and the center of effort is used to calculate moments.
By using sail cases that have been run previously, the hydro solver does not need to wait for an aero iteration. The simulation can run as quickly with accurate, dynamic, sail forces as it would with constant forces. Figure 2 shows the driving force and side force for a range of heel angles using a main+J4 sail configuration.

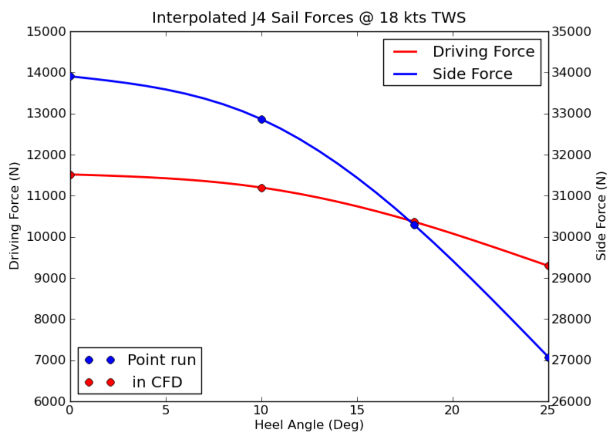

Figure 2: Sail Force

4.2 RUDDER BALANCE

In 6-DoF simulations an unbalanced yaw moment will cause the boat to spin in a circle. To address this problem a rotating rudder is added to the simulation. Varying the rudder angle changes the hydrodynamic moment generated by the boat, which can be made to balance the aerodynamic moment from the sails.

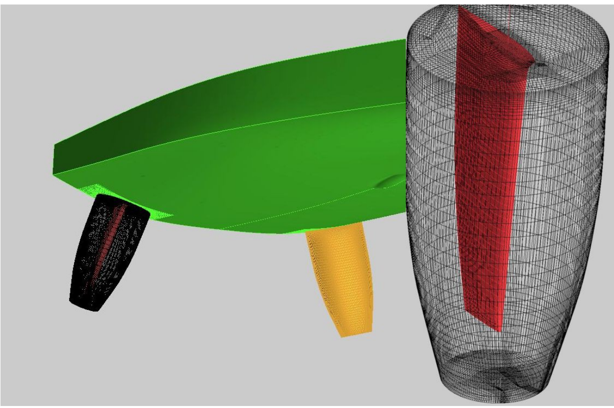

Figure 3: Rotating Rudder Mesh

To incorporate this rotating rudder into 6-DoF simulations a separate mesh is created around each rudder. This mesh is contained within an axis-symmetric volume to allow rotation. A sliding boundary condition is used at the interface with the main hull mesh. Figure 3 shows the rudder mesh for a candidate Open 60 design. Using this sliding interface adds 15% to the computation time and therefore is only applied when needed.
In wave cases a PID controller is used to rotate the rudder and keep the boat on the proper heading. In calm cases, where the external forces on the boat are not varying, the controller can be set to balance the yaw moment directly and equilibrium is reached very quickly. Figure 4 shows the aero and hydro moments and rudder angle for an Open 60 in calm water. The rudder controller is turned on at 11s and the rudder angle changes by 0.8 degrees. This causes the hydro moment to balance the aero moment from the sails.

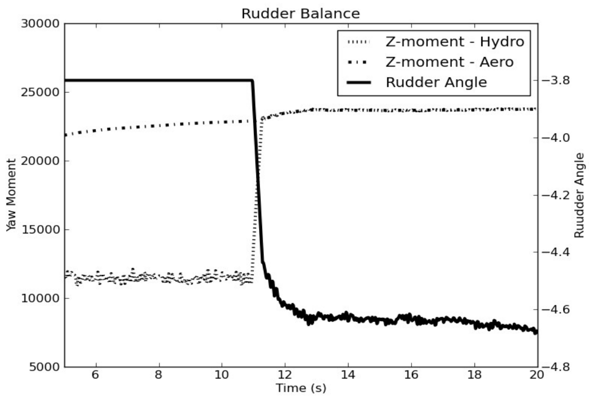

Figure 4: Moments with Balanced Rudder

Including the rotating rudder within the simulation itself, instead of calculating the needed rudder angle afterward, allows the simulation to capture any secondary effects of the new rudder angle. For example, in the case from Figure 4, the lift generated by the keel fin drops by 30% when the rudder angle changes, and the overall boat speed increases by 0.4%. When multi-day ocean races are sometimes won by less than 10 minutes, modeling this 0.4% of boat speed can be incredibly important. For this project, when the candidate boats where compared at this sailing point, the overall boat speed rankings switched when the rotating rudder was included. This highlights the importance of including a balanced rudder in simulations.
Currently this technique has only been used for rudders, but a similar methodology could be used for any control surface. More advanced controllers can be tested and tuned for roll damping or lifting foil applications.

5. 6-DoF IN OPEN 60 DESIGN

With this 6-DoF capability in place a variety of dynamic boat behaviors can be modeled. In the recent design of an IMOCA Open 60, 5-DoF (fixed yaw) and 6-DoF cases were run to investigate specific design features and sailing conditions. In calm water cases the rudder angle was set by the controller to balance the yaw moment. Wave cases in this study used a fixed rudder angle to increase the convergence rate.

5.1 A7 SAIL IN CALM WATER

Initial 2-DoF simulations for the Open 60 showed that the boat was burying the bow when reaching at high speeds using the A7 sail. To fix this problem a new A7 was designed with a lower center of effort and a set of 5-DoF cases was run to check the new trimming behavior. True wind speeds between 22 and 30 knots were tested at 135 degrees of true wind angle. Results showed that at 30 knots with the redesigned sail the bow of the boat just touches the water as intended.
Eight cases were then run with this new sail to compare the reaching performance of two hull design candidates. Figure 5 plots the boat speeds at two wind angles and two wind speeds. A cross-over in relative performance between the two hulls is seen at 134 degrees of true wind angle for the lower wind speed and 136.5 degrees for the higher wind speed.

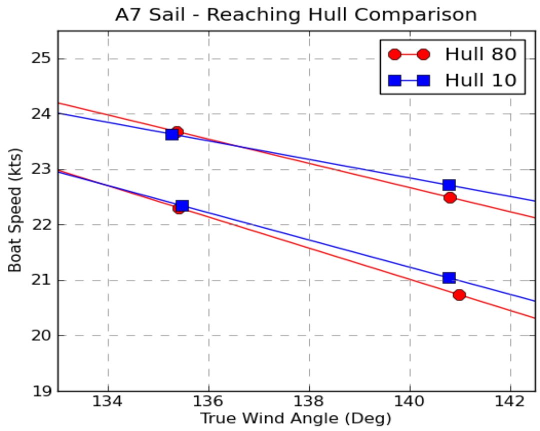

Figure 5: Reaching Speeds

5.2 FORWARD WATER BALLAST

Anecdotal evidence from sailors suggested that increasing the forward water ballast could help the performance of the Open 60 in some upwind conditions, specifically conditions with rough seas and relatively light wind. The design team was skeptical of this claim and wanted to run tests to substantiate or refute it.

A condition with 20 knots of true wind speed and 2.4m waves was tested with single and double forward ballast configurations. The forward water ballast was doubled in the second simulation by changing the center of gravity, mass, and inertia tensor of the boat. Figure 6 shows the boat speed with each ballast configuration. The double forward water ballast did not perform better than the single ballast condition and it was decided to keep the single ballast configuration.

It should be noted that these cases used the same sails and sail trim. With the added weight of the double forward water ballast the boat becomes slightly more stable and could possibly use a more powerful sail trim.

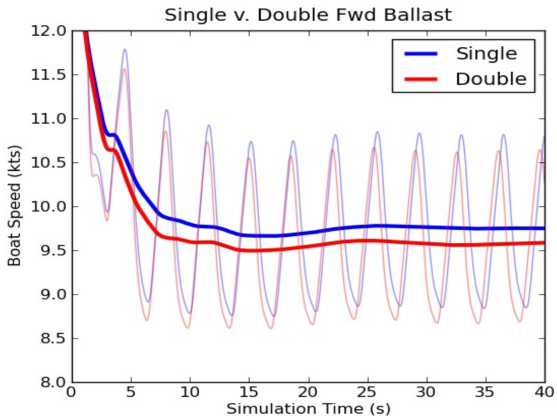

Figure 6: Ballast Comparison

5.3 REACHING IN TRAILING WAVES

Reaching in large ocean waves is one of the most important sailing conditions to consider in Open 60 design. Often in these conditions the waves are overtaking the boat and the hull surfs down each wave, as shown in Figure 8. Simulating cases with opposing fluid flow and wave directions can be a challenge.

To accommodate trailing waves in this study the inlet boundary condition was extended behind the boat and the outlet was confined to the rear left corner of the computational domain. The wave height and velocity is normally prescribed at the inlet; extending the inlet behind the boat allowed the waves to originate behind the boat and propagate forward.

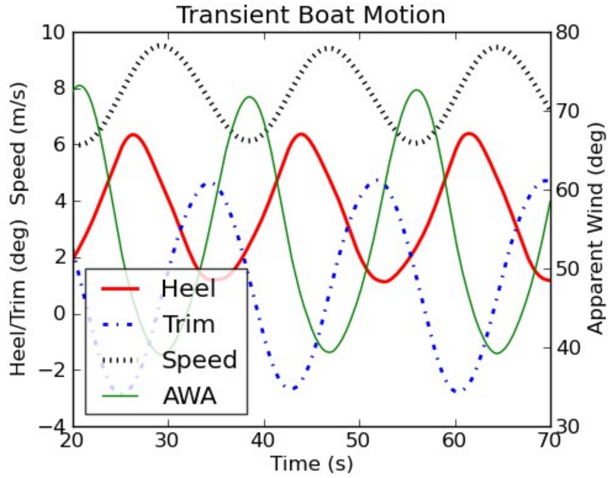

Figure 7: Open 60 in Trailing Waves

Figure 7 shows the boat motion results from a case in waves typical of the southern ocean. The boat accelerates as it surfs down each wave which causes the apparent wind angle to vary by more than 30 degrees. This exceeds the normal range of apparent wind angles that can be reliably tested for a downwind sail geometry. This necessitated using constant sail forces that represent the average orientation and velocity of the boat.

The wavelength used in this case was 169m with a wave height of 3.8m resulting in a wave speed of 31.6kts. The average boat speed of 15.4kts and wave direction of 148 degrees gave an encounter period of over 17s. It was necessary to run for multiple periods for the boat motion to stabilize. This lead to a very long overall simulation time. The case ran for 5 days on 16 computational cores representing a total simulated time of 77 seconds.

This condition was run for a second final candidate hull and, combined with upwind wave and calm water data, was used to select the final hull shape.

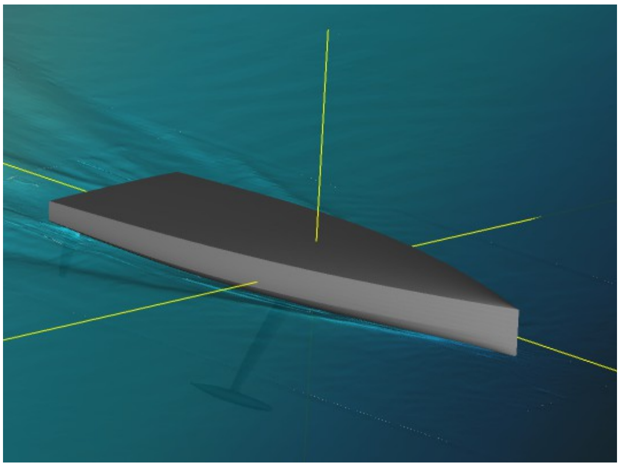

Figure 8: Surfing Down A Wave

6. CONCLUSIONS

In recent years, simulations using RANS codes have surpassed towing tank tests in accuracy, capability, and ease of use. This combined with detailed results and advanced flow visualization techniques gives designers a much greater insight into dynamics and flow characteristics. 6-DoF simulations are another valuable tool helping designers get the most performance out of their boats.

We currently use 6-DoF simulations to assess the balance of racing designs and analyze performance differences between design candidates in specific sailing conditions. In the future we see this methodology eventually replacing the classic VPP.

7. REFERENCES

1. AZCUETA, R., 'Computation of Turbulent Free-Surface Flows Around Ships and Floating Bodies', *PhD Thesis, Technical University Hamburg-Harburg*, 2001.

2. AZCUETA, R., 'RANSE Simulations for Sailing Yachts Including Dynamic Sinkage & Trim and Unsteady Motions in Waves', *High Performance Yacht Design Conference, Auckland*, 2002.

3. AZCUETA, R., 'Steady and unsteady RANSE simulations for planing crafts', *FAST Sea Transportation, Ischia, Italy*, 2003.

4. AZCUETA, R., 'Steady and unsteady RANSE simulations for Littoral Combat Ships', *25th. Symposium on Naval Hydrodynamics, St. John's, Canada*, 2004.

5. CD-adapco, 'Star-ccm+ User guide', *version 3.06.006*, 2008.

6. Cape Horn Engineering website, *www.cape-horn-eng.com*

8. AUTHORS BIOGRAPHY

Rodrigo Azcueta is the founder and manager of Cape Horn Engineering, specializing in marine CFD. He has a degree of 'Doktor Ingenieur' from the Technical University of Hamburg-Harburg, Germany. He has pioneered the application of RANS free-surface flows for ships and floating bodies.

Riley Schutt works as a project manager at Cape Horn Engineering, focusing on hydrodynamic development and multi-degree of freedom simulations. He has an S.B. degree in Aerospace Engineering from the Massachusetts Institute of Technology in Cambridge, MA, USA.