THE ROYAL INSTITUTION OF NAVAL ARCHITECTS

CFD APPLIED TO SUPER AND MEGA YACHT DESIGN

R Azcueta and **N Rousselon,** Cape Horn Engineering, Spain

SUMMARY

This paper presents the application of state-of-the-art Computational Fluid Dynamics (CFD) simulations to sailing and motor yachts. It aims to demonstrate that simulations are a good alternative to tank testing for the super and mega yacht industry. The simulations are based on experience gathered over the last few years designing high performance racing boats, especially in America's Cup and Volvo Ocean Race campaigns, the Formula One of the seas. CFD is one of the new technologies that can, as was done in the auto industry, be successfully transferred from the racing environment to general industry. In particular, the super and mega yacht industry can now benefit from the recent advances in CFD. Cape Horn Engineering is a company that specializes in hydrodynamic and aerodynamic CFD for the marine industry.

This paper presents a general yacht design philosophy based on simulations and its application to a typical power boat and a large sailing yacht.

1. INTRODUCTION

The success gained from the application of Reynolds-Averaged Navier-Stokes (RANS) free surface flow simulations to the design of high performance racing boats, especially in America's Cup and Volvo Ocean Race campaigns in recent years, is undeniable.

The results achieved in these campaigns have made aero and hydrodynamicists, and increasingly yacht designers, extremely confident that simulations, when applied diligently, can be a superior alternative to traditional tank or wind tunnel testing.

Simulations developed in the America's Cup environment require rigorous testing and validation to remain 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 traditional 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 inherent error of scaled test results. Enhanced flow visualization and force decomposition give designers a much greater understanding of flow phenomena. This relatively new technology can now be successfully taken from the racing environment and applied to the super and mega yacht industry, and the marine field in general.

Usually super and mega yachts are not designed with highest performance in mind, instead the focus is comfort, ease of use, and aesthetics while maintaining reasonable performance. CFD simulations can successfully address power requirements of motor yachts, balance of sailing yachts, the behavior and structural loads in waves, and wind on deck or other conditions influencing passenger comfort.

In this paper we will describe the underlying numerical methods first, then explain the general CFD based design philosophy for high performance sailing yachts, and finally show a few examples of applications to super and mega yachts.

2. NUMERICAL METHOD

For our simulations we use 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 that includes the latest physical models and solver technology (turbulence models, transition models, cavitation, 6 degree of freedom body motion, integrated unstructured volume meshing polyhedral and trimmed cell approach, etc).

Both codes have shown to be very flexible and efficient to use thanks to user programming (Comet through Fortran or C user coding, and Star-CCM+ through a Java API).

Comet, when using its cell-based numerical capability, is still our tool of choice for highly accurate and fast free surface computations (where small design changes need to be evaluated). Star-CCM+ is better geared towards complex geometries and complex physics.

Numerical computations for marine applications involve the coupling of a RANS flow solver (with free surface capability) to a body motion solver and a mesh motion/deformation solver. The RANS equations are solved using a finite volume approach. Both solvers handle any cell type and topology, including polyhedral elements.

Accurate viscous solutions require refined mesh resolution near the wall. Two turbulence models, k-epsilon and k-omega, along with wall functions, are generally used to decrease computational time, while keeping an adequate accuracy.

Second order accuracy in space (face values and gradients) is achieved through the use of advanced interpolation techniques.

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. Body orientations and dynamics resulting from the forces acting on the body (viscous and pressure forces) are determined by integrating the equations of linear and angular momentum. External forces and moments (i.e. propulsion from sails or propellers) are added during this step. Basic simulations are free to sink and trim (2-DoF), while more advanced (seakeeping or maneuvering) can be completely free (6-DoF with moving control surfaces).

In cases where dynamic results are not needed (i.e. steady state simulations in calm water), several techniques are used to decrease total computational time. Whether in calm water or in waves, we preferably use a single moving mesh strategy when solving for single body motion. The mesh is not deformed but moved. This is a robust (large motion does not degrade mesh quality) and fast method (no deformation overhead) to handle the motion of the boat.

All of our simulations are processed in parallel on four or eight cores; with 436 processing cores in our cluster we can run more than one hundred simulations 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 first campaigns, ABN Amro and BMW Oracle Racing, a thorough validation with tank testing was conducted. In the current campaigns, Ericsson Racing Team for the Volvo and Team Origin for the America's Cup, the entire design is based on CFD simulations.

The following is a description of our design philosophy for the well funded Ericsson Racing Team campaign.

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 in turn used as input for the hydrodynamic simulations. Later we return to aerodynamic simulations in order 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 then calculates the deformed shape of the sail. The new deformed shape is trimmed by the sail designer and put 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 Volvo 70 Class boats present a new, very complex, design problem. Compared to previous America's Cup Yachts there are many more design variables; the Volvo 70'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 30 knots planing and surfing down waves. The boats also 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 changes 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 inclination, longitudinal inclination (sweep), alignment of the keel cant axis (if allowed by the class), 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 analyse and usually it suffices to compare the amount of drag at a given side force to draw conclusions.

In many cases sail and appendage design and modification is 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.

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.

4. APPLICATION TO MEGA YACHTS

4.1 POWER BOAT

The hull shape of this boat is relatively complex, featuring spray rails and backward steps, see figure 1. Figure 2 shows pressure coefficient contours on the hull for a boat speed of 29 knots. Figure 3 plots the pressure coefficient along the hull centerline, typical pressure peaks are visible.

The speed ranges from low speed in displacement mode to high speeds in planing conditions. The dynamic sinkage and trim of the hull, which is of paramount importance in the higher speed range, is accurately captured by the simulations. Trim and sink values, for a range of boat speeds, are shown in figure 4.

Seakeeping simulations were also conducted, with special attention to added drag, slamming, spray and water-on-deck, see figure 5. Boat motion in a range of wave length and direction was evaluated by freeing 2 to 4 DoF.

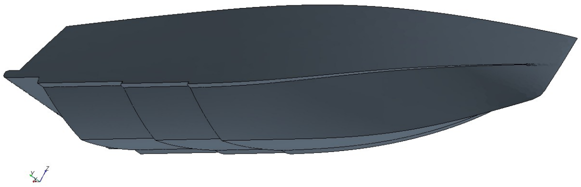

Figure 1: Hull geometry.

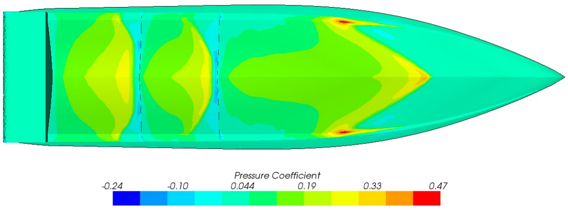

Figure 2: Cp contours on hull at 29knots.

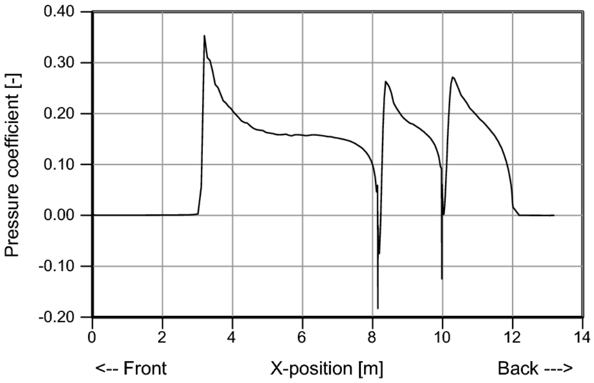

Figure 3: Cp along hull centerline at 29knots.

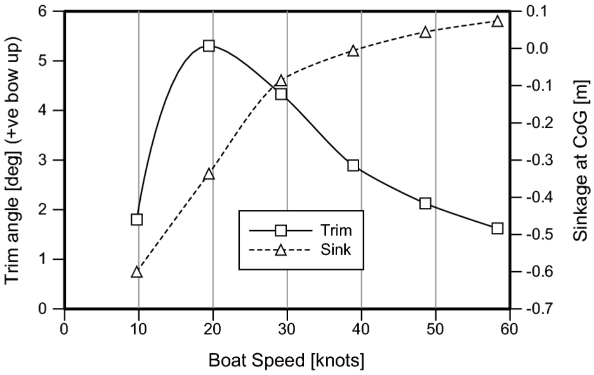

Figure 4: Trim and sink for various boat speeds.

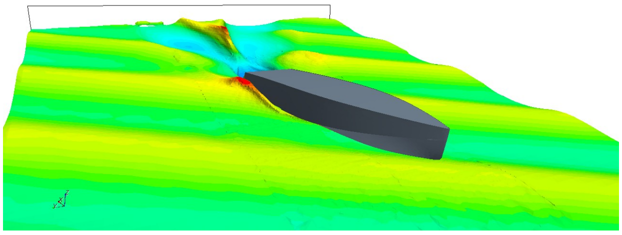

Figure 5: Seakeeping simulation.

4.2 SPEEDBOAT

'Speedboat' is actually the name given to a high performance sailing yacht, conceived to break speed records. She is a 100 foot canting keel yacht, with twin rudders and a very wide and powerful hull shape, see figure 6.

Speedboat was launched in Auckland on 16 April 2008 and that same day underwent the structural test and sail trials, showing a perfectly balanced sail plan and appendages. This behavior would not have been possible without the use of well developed aero and hydro models using CFD, and a detailed VPP.

Today, Speedboat is probably the fastest mono hull sailing boat on the planet.

The simulations conducted for Speedboat aimed at producing a well balanced boat assessing the position of the sail plan and appendages, see figure 7. Furthermore, accurate performance data was obtained to be used as polars on board. Hundreds of simulation points were run, varying heel, yaw, speed and ballast conditions. This together with simulations for the sails were the inputs for the VPP. Finally, simulations in waves were carried out. Figure 8 shows typical results for such a simulation in waves. The wave height is 1 m and the wavelength 30 m at an incidence angle of 30 degrees from the bow. The yacht sails with a heel of 15 degrees at 16 knots. The diagram shows some of the output of the simulation for one wave encounter period. In this case only two degrees of freedom, sink and trim, are free.

Freeing more degrees of freedom makes sense and 6-DoF simulations are currently being developed and validated at Cape Horn Engineering, see [7] for examples. An accurate and robust aerodynamic model is required, it is based on the large library of sail-shapes we have been testing in the past. Changes in rudder angle also need to be modeled; rudder effects are currently included by using a stationary rudder and adding virtual forces from an empirical model; however, they will soon be handled directly in the CFD by rotating the rudder in the simulation.

When looking for a final steady state condition, a control system is put on the rudder for course keeping. Figure 9 shows the motion history for a 6-DoF case: the boat is sailing with a true wind speed of 10 knots and a wind direction of 95 degrees. The rudder control system keeps the boat at a heading of 0.5 degrees.

Figure 6: 'Speedboat' reaching in her maiden outing.

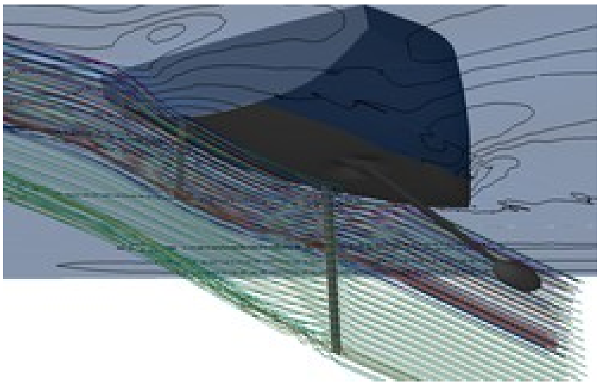

Figure 7: Stream lines and free surface elevation.

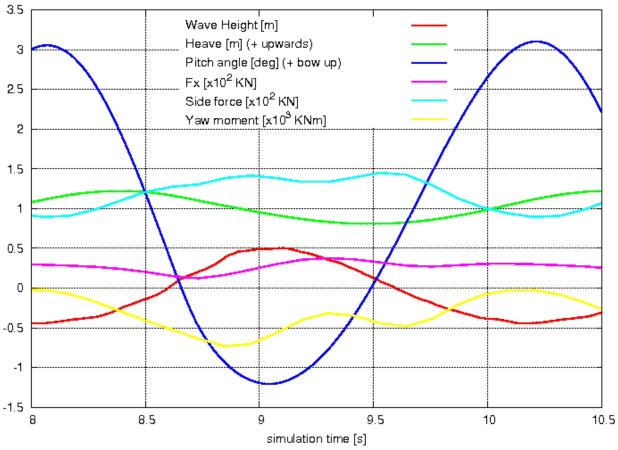

Figure 8: Yacht response in waves.

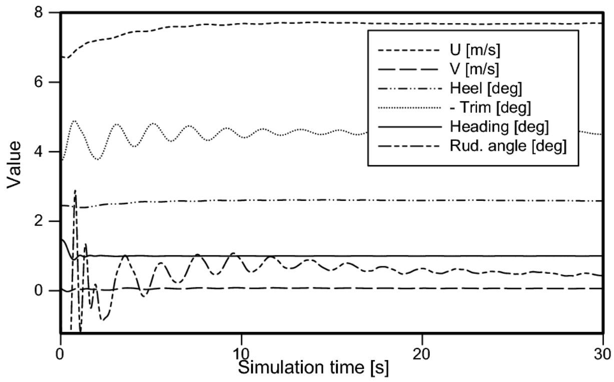

Figure 9: 6-DoF motion history.

5. 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. Using CFD in super and mega yacht design has become a viable and economical alternative to traditional testing.

In the future, we see a trend towards using 6-DoF simulations to assess the balance of sailing yachts and the performance differences between design candidates. This methodology will eventually replace the classic VPP.

6. 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*

7. 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.

Nicolas Rousselon works as a project manager at Cape Horn Engineering, with special focus on method development and aerodynamics. He holds a MSc degree in marine CFD from the University of Southampton, UK.