SIMULATION OF FLOW AND MOTION OF HIGH-SPEED VESSELS

M. Caponnetto, ORACLE Team USA, USA
B. Bučan, M. Pedišić-Buča, Brodarski Institut, Croatia
M. Perić, C. Pettinelli, CD-adapco, Germany/Italy

SUMMARY

The paper describes a numerical method for the coupled simulation of fluid flow and motion of floating bodies with application to high-speed vessels. The main characteristics of the method are: finite-volume approach based on arbitrary polyhedral control volumes which can move and deform during simulation; overset grids to allow arbitrary motion of floating bodies (including capsizing); high-resolution interface-capturing scheme (“volume-of-fluid” approach) which allows simultaneous computation of water and air flow and accounting for wave overturning, trapped air in water and water drops in air; cavitation model based on bubble-dynamics approach that allows prediction of vapour production around body parts where pressure drops below saturation level and bubble collapse in zones of higher pressure. Two application examples are presented: a validation study with a patrol vessel moving over a wide range of Froude numbers, and a demonstration of practical use of simulation to design and optimize the winning boat for the America’s Cup 2013.

1. INTRODUCTION

High-speed vessels move so fast that it often becomes important to simulate air flow as well as water flow. In the past, different approaches and software tools were used for hydro- and aerodynamics analysis, but this does not always lead to an optimal solution: optimization of components alone does not guarantee optimal performance of the system. It is therefore desired to simulate in a coupled way water flow, air flow, and motion of the vessel (ultimately, structural deformation may also have to be simultaneously accounted for, but in this study the floating body is assumed to be rigid).

The advance of computing technology (both hardware and software) has made simulations of performance of complete systems possible. Even small and medium size companies can nowadays afford a computer cluster with hundreds of cores and the modern computational fluid dynamics (CFD) software – as the tool used in this study – comes with a licence that allows the use of an unlimited number of cores. In addition, modern computer-aided-design (CAD) tools provide high-fidelity geometry of the complete vessel with all appendages in digital form, and the grid-generation tools are able to automatically generate a grid suitable for flow simulation without the need to invest a lot of time in manual repair or de-featuring of a complex geometry. With labour-intensive work being relegated to computers, engineers can concentrate on evaluating as large a number of design variants as possible, in order to find one that will satisfy most of desired criteria. While there are automatic optimization tools already available, engineering judgment is still essential since multiple objectives often cannot be optimized at the same time; thus, compromises are the rule rather than exception.

There are several phenomena that characterize the flow around high-speed vessels. Even in still water waves generated by such vessels break or overturn, creating pockets of trapped air in water. Air may also be sucked under the vessel (ventilation is sometimes intentionally provoked or introduced to reduce resistance, but it may also be unwanted). It is often a challenge to distinguish between ventilation as the real flow feature and a numerical artefact (which easily can happen if the grid is not adequate). High-speed vessels often have spray rails and a local mesh adaptation is required in order to correctly predict water separation. Finally, cavitation can happen not only within the propulsion device but also on other parts of hull or appendages where high velocity can locally lead to pressure dropping below saturation level.

When a high-speed vessel moves in waves, it can be excited to large motions and a coupled simulation of flow and motion of the vessel becomes essential. This is often the cause of uncertainties in simulation, since neither the exact position of the centre of gravity nor the exact moments of inertia are usually known. In fact, both can change during motion if people on board, fuel in tanks or other masses in the system also move.

The present paper gives first a short overview of the numerical method implemented in the commercial software STAR-CCM+ from CD-adapco, which can be successfully applied by a knowledgeable user to solve problems associated with high-speed vessels. This is followed by a presentation of a study performed recently at the towing tank facility “Brodarski Institut” in Zagreb, Croatia, where simulations of flow and motion of a patrol boat over a wide range of speeds were conducted first, followed by a detailed experimental study aimed at validation of simulation results. Finally, the success story of the ORACLE Team USA is used as a practical demonstration of the use of simulation to design and then optimize.
an extreme vessel for which past experience did not exist and no experimental studies had been conducted.

2. NUMERICAL METHOD

The software used in this study is based on a finite-volume (FV) method. The conservation equations in integral form (with appropriate initial and boundary conditions) serve as the starting point. The equations will not be reproduced here – the reader is referred to earlier publications in which the discretization and solution method is described in detail [1-3]. Here only a brief description of the main features of the method will be given.

The equations solved (conservation of mass, momentum, volume fraction of water and water vapour, turbulent kinetic energy and its dissipation rate) contain surface and volume integrals, as well as time and space derivatives. These are approximated for each control volume (CV) and time level using suitable approximations: midpoint-rule for integrals, linear or quadratic interpolation and central differences are the typical second-order approximations employed. After applying discrete approximations, an algebraic equation system solvable on a computer is obtained.

The spatial solution domain is subdivided into a finite number of contiguous control volumes which can be of an arbitrary polyhedral shape and are typically made smaller (i.e. locally refined) in regions of rapid variation of flow variables. In studies involving free surface flows, trimmed hexahedral grids (which are automatically generated by the software from the imported CAD geometry) are usually used since they allow anisotropic local refinement, i.e. cells can be refined in vertical direction alone to resolve free surface where lateral refinement is not needed. The locally refined hexahedral grid (made of Cartesian cells) is trimmed by a surface parallel to wall; between wall and this surface, a layer of prismatic cells is introduced to better resolve boundary layer. As will be shown later, this layer of thin cells is also needed to resolve the water layer rising up the hull. Figure 1 shows an example of a trimmed grid with a prism layer around the free surface in bow region of a patrol vessel; the first cell near wall has a thickness of the order of 1/1000th of hull width (typically of the order of a millimetre for a full-size vessel).

Flow around high-speed boats is turbulent and therefore a turbulence model must be activated when solving the Reynolds-averaged Navier-Stokes equations. Over 20 variants of turbulence models are available in STAR-CCM+, from two-equation eddy-viscosity type to full Reynolds-stress model that requires the solution of 7 additional equations. For most studies concerned with forces exerted by fluid on the moving body, the standard versions of k-ε or k-ω turbulence models perform well enough; they were also applied in this study. When the flow structure in the wake is of interest, especially when wall curvature and secondary flow effects are significant, Reynolds-stress model may be required. Ultimately, the large-eddy simulation (LES) or detached-eddy simulation (DES) type models are also available for high-fidelity transient flow simulations, which are usually used for detailed cavitation or hydro-acoustics analysis.

In the equation for volume fraction of water, convective fluxes require special treatment. The aim is to achieve a sharp resolution of the interface between liquid and gas, which requires special interpolation of volume fraction. The method used here represents a blend of upwind, downwind, and central differencing, depending on the local Courant number, the profile of volume fraction, and the orientation of interface relative to cell face; for more details, see [2]. The scheme is adjusted to guarantee that the volume fraction is always bounded between zero and one, to avoid unphysical solutions. The scheme resolves the interface typically within one cell and effectively prevents mixing of liquid and gas; see Fig. 2 for an example from the patrol vessel study.

Cavitation model in STAR-CCM+ is based on bubble dynamics; the equation for volume fraction of vapour has a source term which is proportional to the rate of bubble growth and collapse given by the Rayleigh equation [??]. The proportionality factor is derived based on the assumption that water contains a given number of homogeneously distributed seed bubbles per unit volume (typically of the order of 10^12 bubbles per m^3) of a known initial diameter (typically of the order of a micron). Thus, vapour is produced when pressure drops below saturation.
level (bubbles grow and volume fraction increases) and it reverts to liquid water when bubbles enter a region where pressure is above saturation level (bubbles are collapsing, volume fraction reducing). Note that there is in general no sharp interface between vapour and water, except in recirculation zones with very long bubble residence time, like in the case of sheet cavitation on propeller blades. Often volume fraction of vapour increases and decreases smoothly over a certain volume zone and may not reach 100%.

When the motion of a floating body is also computed, the iteration loop within each time step is extended to allow for an update of body position. The equations of body motion are solved to predict the position at the new time step, based on flow-induced forces and moments; the grid within the flow domain is adjusted in every iteration of one time step to fit the new body position. As the iterations converge, changes in both body position and flow-induced forces become negligible. The solution method is thus fully implicit and fully coupled, allowing adjustment of time step to the desired accuracy rather than the stability criteria, as is necessary when explicit coupling is used. When a nearly steady-state solution with a constant trim and sinking is expected, first-order discretization in time and a limited number of iterations per time step (between 1 and 5) can be used to faster reach the steady state, using time steps of the order of 1/100th of the time required by the fluid to pass one hull length.

Grid adaptation to body motion requires special attention. In the case of a moderate motion of a single body, as would occur when a steady-state solution exists and the initial vessel position is well estimated, one can move the whole grid as a rigid body with the vessel. In this case, one has to ensure that the free surface remains within the grid of the same fineness in the vertical direction, meaning that the grid needs to be kept fine in a wider zone than would be the case if the grid was not moving. For somewhat larger, but not extreme motions, grid morphing can be applied: the grid nodes at body surface move with the body, those at outer solution domain boundary remain fixed, and the nodes in between are moved by a suitable algorithm to adjust to body motion. The third possibility is to use overset grids, where the background grid is adapted to the free surface, incoming waves and outside boundaries (like shore or harbour walls), while overlapping grids are attached to floating bodies and move with them without deformation. In this case the grid quality near body wall (where accuracy requirements are the highest) does not change as is the case with morphing, which usually produces largest grid deformations near moving body. The grid motion is also easier to handle with overset grids, but the solution method is less efficient in parallel processing since the solution domain is split into two regions and the neighbourhood conditions keeps changing during body motion. However, this is the only approach that allows unlimited motion (including capsizing) to be accounted for and when large motions are expected, it is the method of choice. This approach is almost always used when vessels move in waves.

3. VALIDATION STUDY: PATROL BOAT

At Brodarski Institut (Shipbuilding Institute) in Zagreb, Croatia, a towing tank facility is operated which is often used to study performance of yachts, patrol boats and other types of high-speed vessels. Recently, a combined computational and experimental study of a hull designed by Brodarsky Institut has been conducted. Simulations were performed first in full scale, for a wide range of Froude numbers; afterwards, experiments with a model were performed in towing tank, scaled to full size and compared with simulation data. The aim was to verify how well can the performance of such a vessel be predicted by simulation when no information from physical testing is available. Table 1 shows the particulars of the hull. The speed was varied so that the Froude number based on displacement volume varied between 2.13 and 4.25, and the Reynolds number based on length of waterline varied between 0.72×10^9 and 1.27×10^9. Only heave and pitching motions are considered, so simulations were performed for one half of the hull using symmetry plane as one boundary condition. The solution domain extended around one hull length around the hull and deep-water conditions were assumed. Simulations were performed in a coordinate system attached to the hull, i.e. water and air were flowing through inlet boundary at the nominal boat speed and the boat was not moving in horizontal direction.

<table>
<thead>
<tr>
<th>Length (m)</th>
<th>Breadth (m)</th>
<th>Draught (m)</th>
<th>Displacement (m^3)</th>
<th>Wetted area (m^2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.89</td>
<td>3.65</td>
<td>0.78</td>
<td>13.78</td>
<td>37.85</td>
</tr>
</tbody>
</table>

Table 1: Particulars of the patrol boat hull

Figure 3 shows the computational grid on the hull surface, indicating where the grid was locally refined. Several grids were used to determine the grid dependence of solution at one medium Froude number; for the complete study, the grid with a total of 3.43 million cells was selected for which discretization errors were estimated to be of the order of 1-2%. The maximum size of cells attached to hull and within free surface zone was 25 mm, and the thickness of the first cell in the prism layer next...
The time steps were of the order of 1 microsecond.

Local refinement of grid in the zone where surface wetting begins is very important. Figures 2 and 4 show that, as the hull penetrates water, it creates a very thin water layer that shoots up the wall, until it reaches spray rail and gets deflected from wall. If the grid is not fine enough, this thin layer will be unresolved and the cells next to wall will contain a mixture of water and air rather than pure water. Thus, both density and viscosity of the fluid next to wall will be wrong, resulting in lower shear stress. Also, if the grid at the start of wetting does not resolve the upward-turning of the free surface seen in Fig. 2, air may be dragged under the hull, resulting in even larger reduction of friction resistance (this is often called ‘numerical ventilation’).

![Computational grid and computed distribution of water volume fraction (upper) and pressure (lower) in the cross-section shortly after the begin of surface wetting on hull](image1)

Figure 4: Computational grid and computed distribution of water volume fraction (upper) and pressure (lower) in the cross-section shortly after the begin of surface wetting on hull

![Computed free surface shape at steady state, showing the final position of the hull relative to undisturbed free surface and the convergence of resistance, trim and sinkage](image2)

Figure 5: Computed free surface shape at steady state, showing the final position of the hull relative to undisturbed free surface and the convergence of resistance, trim and sinkage

When starting the computation from initial hull position and by imposing constant velocity in all fluid cells corresponding to the vessel speed, forces acting on the hull may initially be extremely high. For this reason, in the simulations the hull was held fixed for 2 s and the motion is released gradually over 1 s to prevent excessive movements due to inadequate initial conditions. As can be seen from Fig. 5, resistance, trim and sinkage stabilise very quickly and the simulation can be stopped after 10 s of simulation time.

The comparison of computed (in full scale) and measured (on a model) resistance, trim and sinkage over the whole range of Froude numbers is shown in Fig. 6. The difference between simulation and experiment is almost everywhere within the experimental uncertainty. This study thus confirmed previous positive experience with using simulation to predict performance of high-speed vessels and their propulsion systems [4-7]. Thus, many preliminary studies can be performed using a ‘virtual towing tank’ in simulation, reducing the number of experiments to the minimum necessary to validate the final simulation result.

![Comparison of predicted and measured resistance (upper) and trim and sinkage (lower) as functions of Froude number based on displacement volume](image3)

Figure 6: Comparison of predicted and measured resistance (upper) and trim and sinkage (lower) as functions of Froude number based on displacement volume

4. **APPLICATION: AMERICA’S CUP BOAT**

Many engineers working on design and optimization of various vessels have already gained enough experience with CFD that they trust simulation at full scale as much as they trust experiment at model scale. Since the extrapolation from model experiment to real-life operation at
full scale involves empirical parameters that require prior knowledge based on similar hulls, the approach becomes difficult when a completely new design is involved and no previous experience with something similar exists. Also, one can never achieve full similarity between experiment and simulation (both Froude and Reynolds number cannot be matched, not to mention scale effects on transition, the effects of geometrical precision and other uncertainties). The ORACLE Team USA had experienced CFD-engineers on board and decided to work solely with virtual towing tank using STAR-CCM+ for simultaneous computation of water and air flow as well as vessel motion. Indeed, over the whole 3 years campaign period, the only experiments were the actual races with the current boat. The race experience was then passed on to the simulation team whose task was to understand the observed behaviour, propose design changes, test the changes in simulation for performance improvement and suggest implementation of changes on the actual vessel.

Figure 7: Testing of different designs for jib and wing end plates to minimize aerodynamic drag of the boat: computed friction coefficient distribution for two different geometries

During the three years of the campaign, the team has performed numerous simulations in the same way as would have been done experimentally in a towing tank. This included optimization of the shape of hull and all appendages, with the advantage that in simulations many more design ideas were investigated than could have been possible in experiments.

The key to boats speed has been the aerodynamics design. Sailing upwind at 25 knots introduces a very high apparent wind speed, so aerodynamics performance of the complete boat structure becomes essential. Wing tip and end vortices are known to cause problems and the team worked hard to minimize their effects by introducing winglets and end plates. Many different versions were studied in simulations; Fig. 7 shows two examples. It was important to look not only at the integral values like drag and lift coefficients but also at the flow features and the local distribution of skin friction, as shown in Fig. 7. The cross beams connecting the two hulls have been streamlined as proper wings (including camber and twist) to control their drag and lift. It was essential to consider the whole system rather than studying each component alone, since the interaction between various parts under different sailing conditions is very strong and can change to a large degree. Figure 8 shows the boat in sailing condition, with one hull out of water; the end plates under jib and wing and the streamlined beams can also be clearly seen.

Figure 8: Top view of the boat in race, showing jib and wing end plates and streamlined beams connecting the two hulls

The virtual towing tank in simulation is at the same time the cavitation tank – something that is normally not possible in experiments, where special, much smaller cavitation tanks are used for cavitation studies. In the current case, many simulations were conducted such that all effects including cavitation were accounted for simultaneously; especially towards the end stage, simulation of cavitation effects became an important aspect of team’s work.

Figure 9: Torpedo at the rudder-elevator-junction (to reduce cavitation) and interceptor at hull stern (to control the pitch)
For example, a torpedo-junction between rudder and elevator has been added to reduce the adverse effects of cavitation just before the turning point in the race, see Fig. 9. Another design change that was introduced during the last week of the race was the addition of a wedge (interceptor) at hull transoms to control the pitch in the transition between displacing and foiling mode of sailing (also shown in Fig. 9).

Simulations were not only used to optimize the boat design; it was also important to provide the so called ‘wing and sail targets’ to the sailor, a table of optimum trims (camber, twist etc.) as functions of the wind speed and targets. Equipped with this data generated in flow simulations, the team could better control the boat performance in different situations during the race. In particular, one of the key improvements of the last days of the race has been the structural change of the wing that allowed to properly load the wing to the targets (more load toward the bottom and the lowest flap). The simulation team interacted closely with the sailing crew, reacting to their observations and needs as fast as possible.

Figure 10 shows a plot from a simulation of boat’s motion in waves. Again, the complete system – including the crew members in realistic positions (although static) – has been included in the numerical model. The ability of the team to perform a large number of complex simulations with a fast response to crew’s observations and demands was greatly facilitated by the key features of STAR-CCM+ software: automatic generation of quality grids for complex geometries, fast computation on parallel computers and the possibility to automate the whole process from geometry modification to the analysis of solution. The grids for simulations like the one shown in Fig. 10 contained typically around ?? millions cells and the simulation was run on ?? cores using ?? cluster…

5. CONCLUSIONS

The aim of this paper was to demonstrate the use of simulation of flow and motion of floating bodies in design and optimization of high-speed boats. Commercial software like STAR-CCM+ can nowadays be used to aid engineers in both improving existing designs and testing completely new ideas. A comprehensive study for a patrol vessel provided a convincing validation of simulation, with computed resistance, trim and sinkage being not only qualitatively but also quantitatively as accurate as the experiment. Especially the fact that simulation can be performed in full scale and that the complete system with all interactions between various components and operating conditions can be accounted for, makes the simulation and indispensable tool for design and optimization of high-speed vessels. This is supported by the experience of the ORACLE Team USA who designed, optimized and analysed the performance of their winning boat using solely simulation and observation from boat’s performance in races. The ability to interact with the crew and perform virtual testing of new ideas at short notice has significantly contributed to team’s victory in America’s Cup.

6. ACKNOWLEDGEMENTS

The authors acknowledge with gratitude the contribution of M. Caponnetto’s colleagues from ORACLE Team USA, Francis Hueber and Michele Stroligo; their support in performing all the simulations was invaluable.

8. REFERENCES