SIEMENS

Ingenuity for life

Siemens PLM Software

Full-scale simulation for marine design

Increasing the value of CFD for flow around ships and offshore structures

Executive summary

Computational fluid dynamics (CFD) is becoming widely used in marine design. Generally it is run at model scale to compare with testing tank validation data. This paper examines some common reservations about running CFD at full scale and aims to encourage full-scale analysis of designs under realistic operating conditions. In many cases, full-scale simulation is more accurate and reliable than alternatives and leads to greater understanding of design performance.

Professor Milovan Perić

Contents

Abstract 3
Wall boundry layer.. 4
Reynolds number effects on flow features 7
Scale effects on cavitation 9
Scale effects on energy saving and
flow control devices... 11
Full-scale validation data 12
Conclusion 13
References 13

Abstract

Flow simulation based on solving the Reynolds-averaged Navier-Stokes (RANS) equations has become commonplace in many industries, including shipbuilding and offshore engineering. This approach has become known as CFD, although strictly speaking, simulation methods based on potential flow theory also belong to CFD. Although in many other industries CFD has, to a large degree, replaced experiments on models (for example, in automotive and aerospace engineering), in the maritime industry CFD is still not trusted in equal measure for full-scale applications.

There is a lot of evidence from comparisons between CFD and measurement data at model scale that simulation can be used to reliably predict hull resistance, propeller thrust, cavitation pattern, added resistance in waves, wave-structure interaction, etc. However, when it comes to applying CFD at full scale, there are still many reservations. Some reservations are based on the belief that at such high Reynolds numbers, wall boundary layers cannot be handled accurately enough in a simulation; other doubts exist because there is limited test data from full-scale measurements that can be used for validation.

The aim of this paper is to encourage the use of CFD for simulations at full scale. We are confident that the accuracy of properly conducted CFD prediction at full scale is no worse than at model scale, and the reliability of the results is no worse than the reliability of extrapolation from model-scale experiment to full scale. In many cases, full-scale prediction is more accurate and reliable. The reasons for this confidence are based on experience with Siemens' Simcenter™ STAR-CCM+™ software.

Wall boundary layer

The quality of numerical solution of RANS-equations depends strongly on two factors: 1) The quality and resolution of the numerical grid, and 2) the turbulence models and treatment of wall boundaries. Since Simcenter STAR-CCM+ provides all the necessary tools and features required for generating high-quality grids, this issue will not be dealt with in detail. Simcenter STAR-CCM+ also provides a large number of RANS turbulence models (several variants of both k-ϵ and k-ω eddy viscosity models, including variants for modeling transition from laminar to turbulent state; several variants of Reynolds-stress model), as well as large-eddy simulation (LES) type models and the combination of the two approaches, detached-eddy simulation (DES) variants. For wall treatment, the so-called low Reynolds number (low-Re) approach (for which the grid has to resolve the boundary layer down to the viscous sublayer), high Reynolds number (high-Re) wall functions and the combination of the two approaches (so-called "all-y^+ wall treatment") are available.

Whenever possible, it is good to resolve the boundary layer so the low-Re approach can be used. A suitable mesh requires many prism layers near the wall (typically around 20), and if a high cell count is to be avoided, these prism layer cells must be thin with a high aspect ratio. Such thin cells are not problematic on a flat surface, but where the wall is curved, the grid needs to be refined in a wall-tangential direction for two reasons: 1) The cells become warped when grid lines are not aligned with wall curvature and the warpage should not be too large, for both accuracy and stability reasons, and 2) along a curved wall, both velocity and pressure vary significantly in wall-tangential direction and this variation needs to be adequately resolved. Figure 1 shows a thin, warped cell from a prism layer at the rear part of a container ship at model scale, when the grid was fine in wall-normal direction (leading to $y^+ \approx 1$), but not fine enough in wall-tangential direction: cell thickness 0.03 mm, lateral cell size approximately 18 mm, aspect ratio around 650; 20 prism layers, in total about 2 million cells for half a ship model. When the grid is refined in wall-tangential direction such that the aspect ratio is reduced to around 200, the warpage is reduced sufficiently, leading to a grid of acceptable quality – but the number of cells is increased to approximately 10 million.

At full scale, the Reynolds number is much higher than in model scale and if one wants to resolve the viscous sublayer (which is relatively thinner as the boundary layer thickness scaled by ship length is smaller than at model scale), the aspect ratio in the near-wall prism layer becomes inevitably higher if the wall-normal cell size is of the same order relative to ship length. For reasons mentioned above, a substantial grid refinement in wall-tangential direction would be required in order to reduce the aspect ratio to acceptable values (200 or less), leading to a large number of cells (about 100 million). Although in

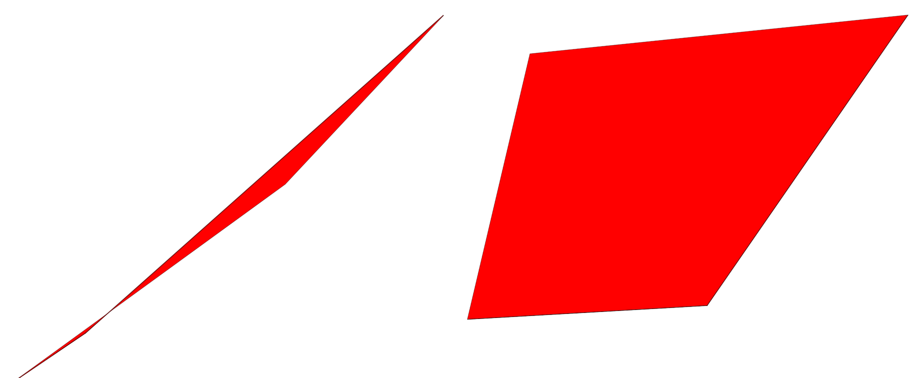

Figure 1: Warped prismatic cell next to a curved wall viewed from two directions.

some industrial applications this number of cells is not unusual any more (given the trend, within a few years grids of this size will become commonplace), it is still considered too large for simulations in the design stage or for optimization purposes.

Another approach to wall treatment is to use the so-called wall functions: The grid only resolves the logarithmic part of the boundary layer, leaving the viscous sublayer and the buffer layer unresolved (see figure 2 for a typical velocity profile in the wall-normal direction across a boundary layer). Using some additional assumptions (which are, strictly speaking, valid only under some conditions and certainly do not apply along the entire ship hull), one can compute the wall shear stress based on the variable values at the center of the cell next to the wall. Experience has shown that good results in model scale studies (for which there is enough validation data from experiments) can be obtained if the dimensionless distance of the first cell center from the wall, the so-called y^+, is between 50 and 100. For example, at Hyundai Heavy Industries hull resistance has been computed on about 200 hull shapes at model scale and compared with experimental results; in most cases the predicted values differed from measured ones by less than 2 percent.

Based on the above discussion, wall functions are the method of choice for full-scale analysis since the resolution of the viscous sublayer (that is, having a y^+ value of 1 at the center of the first prism layer) would require excessively fine grids. However, even achieving a y^+ value of between 50 and 100 results in high aspect ratio and warped cells. Simply scaling up the grid used for model-scale analysis leads to y^+ values at the first cell center around 10,000, which appears too high. Thus, many people believe that CFD in full scale is too costly because one needs extremely fine grids in order to obtain y^+ values at the near-wall cell center in the range that is considered reliable based on model-scale studies.

Let us do a simple analysis to see whether one really needs the same y^+ values in both full scale and model scale when using wall functions in order to achieve the same reliability of results. Note that in maritime engineering, model experiments are conducted by enforcing the same Froude number,

$$\mathrm{Fn} = \frac{U}{(gL)^{1/2}}$$

Here U is ship speed, g is gravity acceleration and L is ship length. Thus, for a ship model scaled by a factor s, the length of the model is related to the full-scale ship length as:

$$L_{\mathrm{mod}} = \frac{L_{\mathrm{full}}}{s}$$

The ship speed scales as:

$$U_{\mathrm{full}} = \mathrm{Fn}\ g^{1/2}(L_{\mathrm{mod}}s)^{1/2} = U_{\mathrm{mod}}s^{1/2}$$

The Reynolds number is thus much larger in full scale than in model scale (with the same fluids, water and air, one cannot achieve similar Froude and Reynolds numbers):

$$\mathrm{Re}_{\mathrm{full}} = \frac{u_{\mathrm{full}}L_{\mathrm{full}}}{\nu} = \mathrm{Re}_{\mathrm{mod}}s^{3/2}$$

Here ν stands for the kinematic viscosity of water.

Let us now see the consequences of the Reynolds number mismatch for the boundary layer. Since we are only interested in a qualitative assessment, let us consider the simpler geometry of a flat plate. The boundary layer thickness grows on a flat plate as:

$$\delta_x \simeq 0.382x\mathrm{Re}_x^{-1/5}$$

where Re_x is the Reynolds number based on plate length x. The boundary-layer thickness thus scales as:

$$\delta_L^{\mathrm{full}} = 0.382L_{\mathrm{full}}\mathrm{Re}_{\mathrm{full}}^{-1/5} = \delta_L^{\mathrm{mod}}s^{7/10}$$

This means the boundary layer over a full-scale ship is relatively thinner than on a model ship.

The skin friction coefficient C_f scales approximately as:

$$C_\mathrm{f} = \frac{2\tau_\mathrm{w}}{\rho U^2} \simeq 0.074\mathrm{Re}_x^{-1/5}$$

where τ_w is the wall shear stress and ρ is water density. The wall shear stresses on a full-scale and model ship are thus related in the same way as the boundary layer thickness:

$$\tau_\mathrm{w}^{\mathrm{full}} = \tau_\mathrm{w}^{\mathrm{mod}}s^{7/10} = \tau_\mathrm{w}^{\mathrm{mod}}s^{0.7}$$

If the optimal model-scale grid is simply scaled by s to obtain the full-scale grid, what happens to y^+? The y^+ is proportional to the square root of wall shear stress and distance from the wall:

$$y^+ = \frac{u_\tau y}{\nu} \quad \text{where} \quad u_\tau = \sqrt{\frac{\tau_\mathrm{w}}{\rho}}$$

where y is the distance from the wall. By taking the above relation for wall shear stress scaling and the scale s for distance, one obtains:

$$y^+_{\mathrm{full}} = y^+_{\mathrm{mod}} s^{27/20} = y^+_{\mathrm{mod}} s^{1.35}$$

For a scaling factor s = 50, this would mean, for example, if in model scale one has at the first cell near wall y^+_{mod} = 50, for the full scale (if the mesh is simply scaled up by s) one would have y^+_{full} = 9,830. This is indeed too much: If one wanted to achieve y^+_{full} = 50 (the same as in model scale), the thickness of the cell next to the wall would have had to be reduced by a factor of approximately 200, making the aspect ratios larger at full scale than at model scale by the same factor.

However, there is no reason to require the y^+ values to be the same in model and full scale. The logarithmic range in velocity profiles starts at about the same value of y^+ irrespective of the Reynolds number, but it extends to much higher y^+ values as the Reynolds number increases. This is visible from direct numerical simulation (DNS) – computation on grids with time steps that resolve all turbulence features in both space and time. Figure 2 shows velocity profiles in a plane channel at four different Reynolds numbers obtained in DNS simulations.

Although DNS cannot be applied to high Reynolds numbers due to limited computing resources, one can clearly see how the logarithmic range increases up to y^+ = 2,000 for the largest Reynolds number (the Reynolds numbers shown are defined with friction velocity u_τ; the largest one corresponds to the Reynolds number based on channel height and mean velocity of 250,000). Experimental data from the superpipe experiment at Princeton University (including probably the highest Reynolds number flow studied experimentally in a laboratory) shows logarithmic range extending to $y^+ \approx$ 100,000. For a detailed study of the log law, see papers by Wosnik et al. (2000)[1] and Lee and Moser (2015)[2].

The above information suggests it is neither practical nor necessary to require the first computational point next to a wall to be placed at the same dimensionless distance from a wall in full and model scale (for example, $50 \le y^+ \le 100$). It is only important there are the same number of computational points within the logarithmic range. For model-scale simulations, when the Reynolds number is not very large the logarithmic range is short and one should therefore start in the region with lower y^+-values. At full scale, the logarithmic range extends to much higher values of y^+ and one can therefore start with values larger than 1,000. It is quite appropriate to place the same number of computational points within the logarithmic range in model and full scale, and to distribute them in the same way relative to the total width of the logarithmic range. As shown in the above example, the logarithmic range in full scale extends to y^+ values about two orders of magnitude higher than in model scale.

If one takes a point in the same relative position within the boundary layer in full scale and in model scale, the ratio of y^+-values would be:

$$y^+_{\mathrm{full}} \simeq \left(\tau_{\mathrm{w}}^{\mathrm{mod}} s^{7/10}\right)^{1/2} \delta_L^{\mathrm{mod}} s^{7/10} = y^+_{\mathrm{mod}} s^{21/20} = y^+_{\mathrm{mod}} s^{1.05}$$

In the example used above (s = 50), for y^+_{mod} = 50 the corresponding full-scale value would be y^+_{full} = 3,040. Thus, for a similar distribution of computational points within the boundary layer, the near-wall cell should be 3.23 times thinner in full scale than the scaled up cell size from model scale. For a scaling factor s = 30, the ratio would be 2.77. This means one needs two to three more prism layers in full scale than in model scale so the scaled-down, near-wall cell thickness from full scale is about three times smaller than the optimal cell thickness in model scale.

Of course, it is not wrong to use grids with lower values of y^+ in full scale as long as they fall within the logarithmic range. For example, a grid that resolved the viscous sublayer in model scale, if scaled up to full-scale size, would lead to y^+ values in the range around 200, which would be perfectly suitable provided the grid is sufficiently refined in tangential direction over curved surfaces. With coarser grids, higher y^+ values should be used in order to avoid excessive cell aspect ratio and warpage.

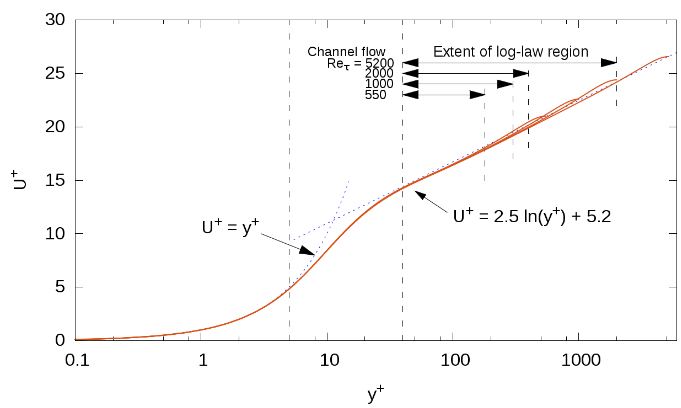

Figure 2: Shown is the variation of the velocity profile within the boundary layer as a function of the Reynolds number for a fully developed flow in a plane channel simulated by DNS [from various databases].

Reynolds number effects on flow features

With an increasing Reynolds number (either due to increased speed or geometric size), flow features may significantly change. The well-known example is the transition from a laminar to turbulent regime. On a full-scale ship, only a fraction of the bulbous bow is covered by a laminar boundary layer. This fraction depends on the smoothness of the surface: roughness leads to an earlier transition to turbulence. In model scale, a quarter of a smooth, newly painted model may be in laminar regime. This makes it practically impossible to use the extrapolation of experimental data obtained in model-scale flow to predict the full-scale flow. The problem is overcome by using tripping: The boundary layer is forced to become turbulent at the "right" bow location, so the majority of the hull surface is under a turbulent boundary layer. However, this approach requires a careful calibration as labs use different tripping mechanisms (from sand paper or trip wires to special patterns of pins with different diameters and lengths) and extrapolation procedures.

Although this approach usually works well for standard ship shapes (for which a large amount of past data and verifications from sea trials exists), every novel shape poses a challenge since the optimal tripping of the boundary layer depends on both the hull shape and the Reynolds number.

In CFD, full scale is less problematic than model scale: One can rightfully assume the entire wall surface is covered by a turbulent boundary layer, which removes the uncertainty related to laminar-turbulent transition. At model scale, simulating flows on parts with independent boundary layer development (such as propeller blades, energy saving devices and other appendages) requires modeling the transition to turbulence rather than applying tripping. Although Simcenter STAR-CCM+ includes models for laminar-turbulent transition, this is usually the weakest link in the chain since transition depends on many factors (like free-stream turbulence, surface roughness, pressure gradient, etc.).

Another important change in flow features occurs with many bluff bodies when laminar separation with a turbulent wake (which is obtained at lower Reynolds numbers) switches to turbulent separation after the boundary layer becomes turbulent before separation. This leads to the so-called drag crises; well-known examples are flows around a circular cylinder and a sphere. Figure 3 shows the qualitative variation of drag coefficient for spheres with smooth and rough surfaces, illustrating the drag crises and its dependence on surface roughness. Similar variation is obtained for cylindrical structures and other bluff bodies.

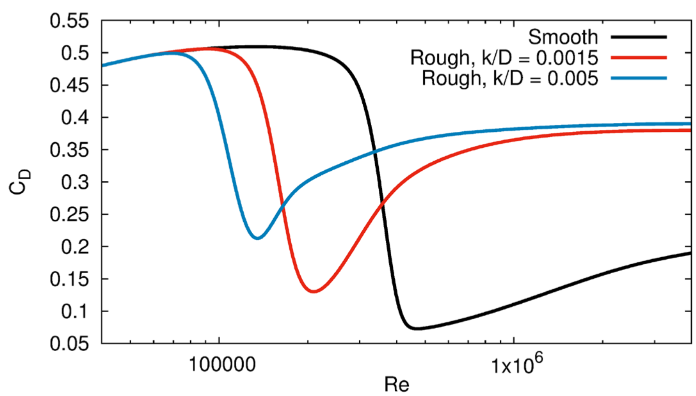

Figure 3: Variation of drag on sphere with Reynolds number, for a smooth surface and various rough surfaces.

If an experiment is conducted at model scale enforcing Froude similarity (because of free surface and wave phenomena), then the Reynolds number will be much smaller than in full scale. If the Reynolds number at model scale is in the subcritical regime (before the drag crises), it is difficult, if not impossible, to find a reliable extrapolation that correctly predicts the state at full scale. One can trigger transition to turbulence sooner by using surface roughness, which moves the drag crises to the lower range of Reynolds numbers so the flow becomes supercritical even at model scale (see figure 3). However, as can be seen in figure 3, surface roughness increases the friction contribution to drag and thus it remains substantially higher than for the supercritical range of Reynolds numbers when the surface is smooth. This makes it difficult to extrapolate from a model-scale experiment to the full-scale flow. In order to predict full-scale properties from experiments at model scale, one needs to calibrate the extrapolation procedures with available full-scale data. The problem is one usually obtains a limited set of data from sea trials with a newly built ship; for offshore structures, there is even less data that is suitable for calibration purposes. Also, different quantities require different calibration: Even if one tunes the extrapolation to match, say, ship resistance, the velocity field in the wake and around appendages will not necessarily scale in the same way.

Even without drag crises, flow features at model and full scale can be substantially different. Flow separation from a smooth surface is generally strongly dependent on the Reynolds number and is difficult to scale up. Both the width and the length of the separation zone can change significantly if the Reynolds number is considerably increased, leading to different wakes behind the body. The drag force is usually also changed significantly. The flow at model scale may not be similar enough to the flow at full scale, making the extrapolation in Reynolds number space difficult.

An example of such phenomena is shown in figure 4. Turbulent flow of air around a square cylinder with rounded front edges is studied at two Reynolds numbers: 500,000 and 5,000,000. The mean flow velocity is in both cases of 12.5 m/s; the Reynolds number is made 10 times larger in the second case by scaling up the geometry by the same factor. In the first case, the width of the cylinder is 0.6 m and the radius of curvature is 5 mm; in the second case the corresponding dimensions are 6 m and 50 mm, respectively. As the Reynolds number increases, the flow pattern changes substantially: At the lower Reynolds number (representing a model-scale experiment), the flow separates at the rounded front edges sooner, forming a larger angle than at the higher Reynolds number (representing the full-scale flow). The consequence is that at model scale the recirculation zone from the separation at the front edges joins the recirculation behind the cylinder, while these recirculation zones are separated at full scale. The drag coefficient is almost twice as large at the lower Reynolds number (1.1) than at the higher Reynolds number (0.66); the maximum velocity around the rounded edge is substantially higher at full scale – 29.1 m/s vs. 23.4 m/s.
This shows clearly that for this type of flow, predicting full-scale flow behavior based on model-scale experiment is extremely difficult.

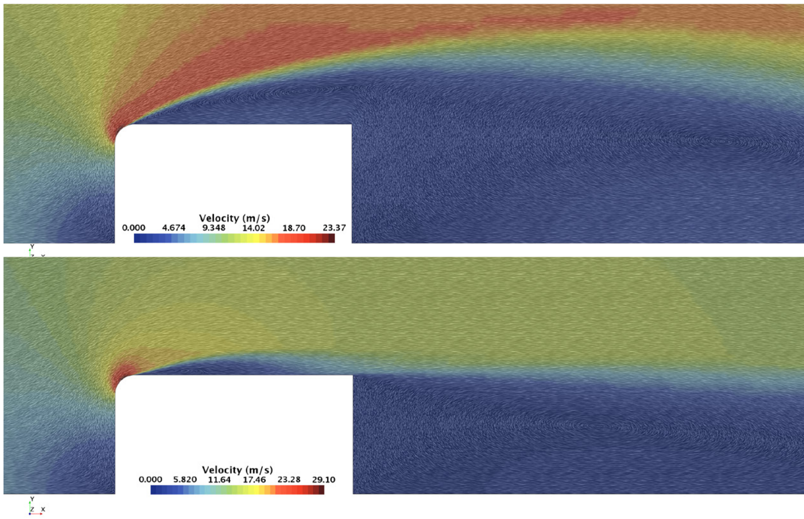

Figure 4: Flow separation on square cylinder with rounded corners: Reynolds number 500,000 (upper) and 5,000,000 (lower).

From the above discussion it is clear the inability to match both Froude and Reynolds numbers remains a significant problem for experimental results, especially when analyzing a novel design (when there is no experience with similar shapes). On the other hand, CFD simulations can be performed at full scale as easily as at model scale. The uncertainties come from the same sources in both cases: 1) grid quality and resolution (discretization errors) and 2) turbulence modeling (modeling errors). The modeling errors are likely to be of the same kind and order in both cases. In order to keep discretization errors of the same order, one needs to refine the prism layers near wall, as outlined in the previous section.

Scale effects on cavitation

Cavitation is another physics phenomenon that requires modeling since we cannot afford to resolve each vapor bubble in the computational grid. The use of a cavitation model introduces additional modeling errors into the simulation. However, experimental analysis of cavitation is also problematic and predicting full-scale behavior based on cavitation tunnel data is not always reliable. As discussed in the previous section, the reason is one cannot obtain complete similarity of model and full-scale flow. Experiments are usually performed in a cavitation tunnel without free surface, while in reality there is an uneven free surface above the propeller. When a ship moves in waves, ventilation can happen at times when the propeller submergence is below critical level; this cannot be simulated easily in a cavitation tunnel. In order to account for the missing free surface, one usually attaches the ship model to the top tunnel wall at a plane that is above the design still water level and also inclined with respect to it; this effectively changes draft and pitch angle. There is no clear formula to specify how to make these adjustments in order to best predict propeller performance in full scale; each towing tank facility has its own empirical procedures.

One of the problems faced in experiments is the geometrical fidelity of model and full-scale propellers. Although real propellers have a diameter of about 10 m, models are usually around 0.2 m in diameter. The curvature of leading and trailing blade edges as well as blade roughness are difficult to precisely reproduce in model scale. The transition from laminar to turbulent flow regime is another problem. However, one of the greatest problems is that owing to the smaller model propeller diameter, it rotates much faster (10 to 20 times) than the full-scale propeller. In addition, the hydrostatic pressure varies around one bar with depth in full scale, while the variation in model scale is negligible. As a consequence, cavitation patterns are totally different in model and full scale. Figure 5 shows the zones in which vapor volume fraction is above 5 percent for the same propeller, obtained from simulations in model and in full scale. In the model scale simulation, all blades are affected by cavitation almost at all times; but in the full-scale simulation, cavitation is present on the propeller blades only during a fraction of a revolution. In simulations, boundary conditions in model scale are the same as in the experimental setup, while in full scale the proper environmental conditions are accounted for as the propeller rotates attached to a moving ship with free surface deformation also being computed. Thus, the only additional uncertainty in simulation comes from the cavitation model: Accounting for a free surface (whether flat or with waves) and ship motion doesn't present problems or excessive additional effort.

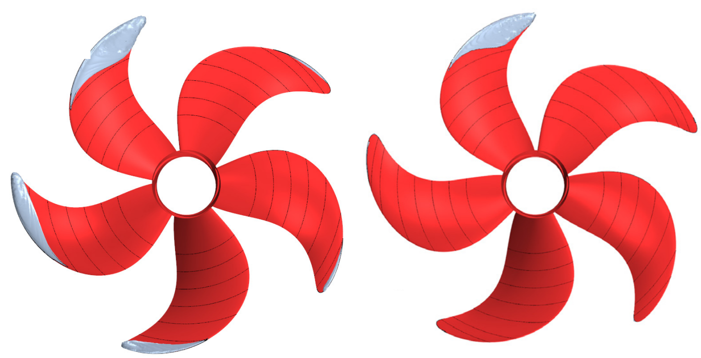

Figure 5: Predicted cavitation pattern on a propeller running between hull and rudder: in a cavitation tunnel at model scale (left) and under free surface at full scale (right).

The modeling errors can only be assessed when the discretization errors are much smaller, and reliable information about the real flow exists. For cavitation on propellers, there is a substantial amount of flow visualization data in both model and full scale, showing cavitation zones. On the other hand, good agreement between simulation and measurement is obtained for non-cavitating flow around the propeller, and grid dependence studies routinely suggest that further refinement does not lead to a significant change in thrust and torque. However, when applying the same grid to the cavitating flow, one does not get cavitation in the tip vortex. Until recently it was believed this was due to a deficiency in cavitation models. Recent studies have shown if the grid is locally refined adequately, the cavitation model in Simcenter STAR-CCM+ correctly predicts the cavitation pattern within the tip vortex. Figure 6 shows the sections through a grid suitable for non-cavitating flow and through a locally refined grid to capture cavitation in the tip vortex.

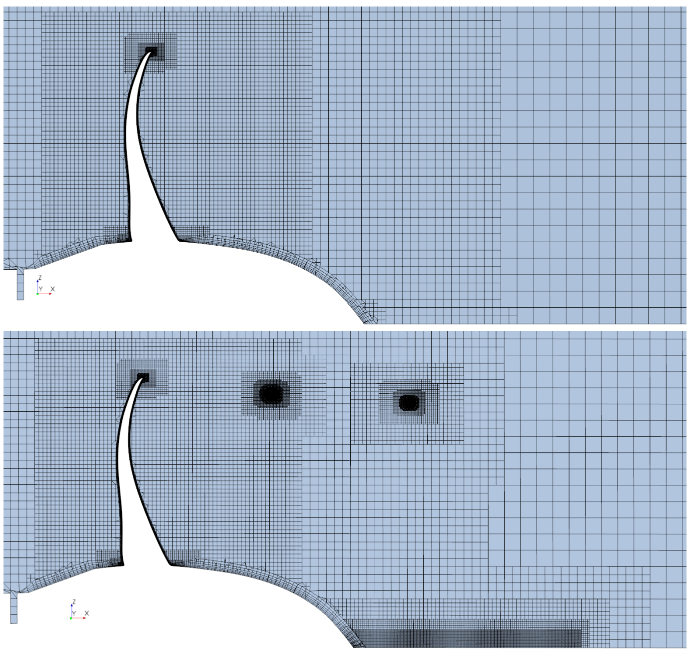

Figure 6: Computational grid used to simulate cavitating flow around a propeller (test case from the Symposium on Marine Propulsors 2011 workshop, smp'11). The upper image shows the usual grid design suitable for non-cavitating flow, and the lower image shows the grid designed to resolve tip vortex by a substantial local refinement.

Without a substantial local grid refinement, cavitation in the tip vortex is not captured, as can be seen in figure 7. However, if the grid is locally refined sufficiently within a narrow spiral zone identified by a threshold based on the magnitude of vorticity, the tip vortex cavitation comes out nicely. In the case in figure 7, the cell size within the tip vortex is of the order of 0.2 mm (for a model propeller with a diameter of D =250 mm, that is the cell size is smaller than D/1,000). In full scale, the diameter of tip vortex is larger so the grid spacing will not have to be that small, but it certainly has to be of a similar order relative to propeller diameter.

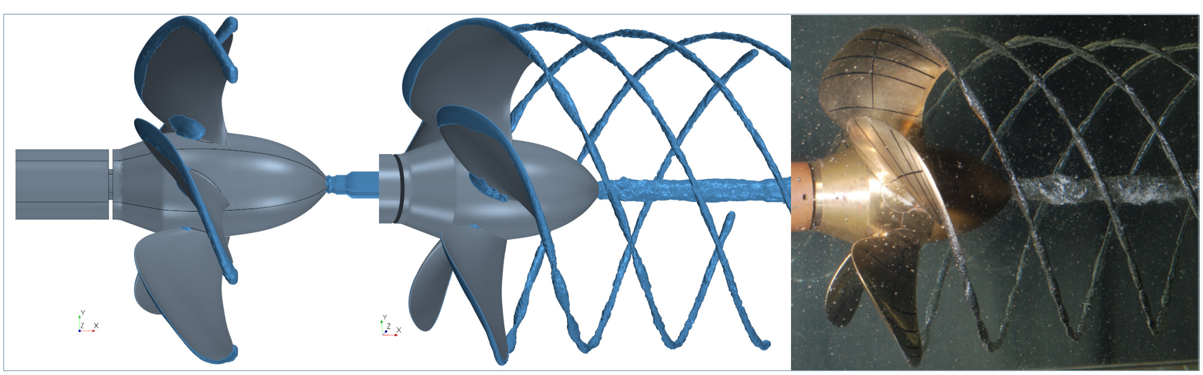

Figure 7: Predicted extent of cavitation (represented by the iso-surface of vapor volume fraction 0.05) in a flow around a propeller, computed using the two grids shown in figure 6 (left and middle) compared to visualization from experiment
(right: courtesy of SVA Potsdam).

Scale effects on energy saving and flow-control devices

Energy saving and flow-control devices are usually small parts relative to ship size; other appendages on ships or small geometry details on offshore structures are also often several orders of magnitudes smaller than the main structure (like risers, pipes, cables and other small parts of an offshore platform). These geometrically small features can have a significant effect on the flow, and if correctly designed, can lead to energy saving, preventing vortex shedding and structural oscillations, etc.

The problem with such small parts is when they protrude into the flow, the Reynolds number based on their characteristic length may become subcritical at model scale. For example, cylindrical parts of an offshore structure may develop laminar separation with a large turbulent wake at model scale (as is characteristic of subcritical flow around a circular cylinder), while at full scale the boundary layer would become turbulent before separation and a much smaller recirculation zone would result. If a fin, strut or other foil-like structure is supposed to be aligned with the flow, and alignment is achieved at model scale, the flow may separate from the suction side at full scale because the direction of the on-coming flow may change as the Reynolds number increases. It is therefore difficult to analyze, and especially to optimize, energy saving devices and similar small parts using scaled-down models. Although experiments may be difficult to conduct at full scale, for the simulation the effort is nearly the same, irrespective of the size of flow domain. Indeed, problems with several designs not performing in full scale as expected based on model-scale experiments have been solved using full-scale simulations.

One of the greatest advantages of simulation over experiments is it always provides complete information about the flow: Even if you just want to compute the drag of a body, you can obtain the velocity and pressure fields for the entire solution domain, along with the information about turbulence and any other derived quantity, like vorticity or other vortex-identification criteria. The ability to visualize the flow (nowadays one can even use virtual reality tools) often helps the engineer to understand the cause of a problem or get an idea how to improve the product. Figure 8 shows the visualization of flow in the vicinity of walls using shear stress vector distribution. Figure 9 shows vortical structures behind a propeller and their interaction with a rudder. Such full-scale flow analysis is essential if you want to design and optimize energy saving devices. Some devices currently on the market were invented after a user analyzed data obtained in full-scale flow simulation.

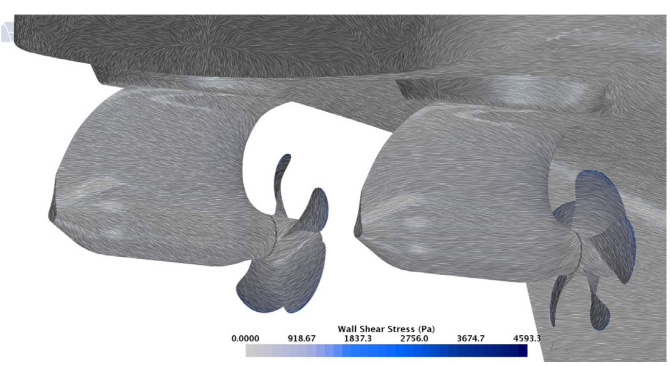

Figure 8: Predicted distribution of shear stress on wall surfaces of a full-scale ship equipped with two Siemens eSIPOD drives. This example shows simulation of self-propulsion with a specified propeller rotation rate.

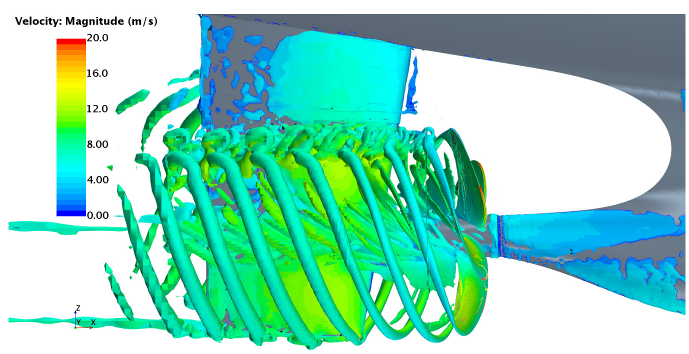

Figure 9: Predicted iso-surface of Q-criterion (which is used to identify vortical structures in the flow) around propeller and rudder, colored by the vorticity magnitude.

Full-scale validation data

The amount of data obtained on full-scale maritime structures is limited. Lloyd's Register recently made data publically available from one of their many full-scale measurements for a medium-size ship. In November 2016, Lloyd's Register organized a workshop at which more than 20 companies presented comparisons of simulation results with this measurement data. The task was to predict the self-propulsion point at three operating conditions. Around 60 percent of submissions for each test case were produced using Simcenter STAR-CCM+. The results which came closest to experimental data (less than 1 percent difference) were also obtained using Simcenter STAR-CCM+. More information about the workshop and the results can be found in the proceedings[3]. The workshop demonstrated if simulation is performed by capable engineers using state-of-the-art CFD tools, they can predict ship performance at full scale. More full-scale data is desirable in order to raise the confidence in the usefulness of simulation in predicting the full-scale flow. Even if experimental data is limited, it can still be useful to validate the CFD results. For example, data such as ship speed as a function of propeller revolution and sea state is routinely collected during sea trials before delivery of a new vessel. This could be helpful for use in validation.

It has already been recognized – both experimentally and in simulation – that a vessel optimized under calm sea conditions may not be optimal when operated in waves. It is therefore important not only to perform the simulation in full scale, but also to include realistic operating conditions in the analysis. As shown in figure 10, the wave pattern around a vessel changes substantially even when only small-amplitude waves are present. This affects ship resistance and thus the speed loss (in both simulations, the propeller rotation rate is the same). Knowing in what kind of environment the vessel will usually operate makes it possible for engineers to take the operating conditions into account when designing and optimizing the vessel.

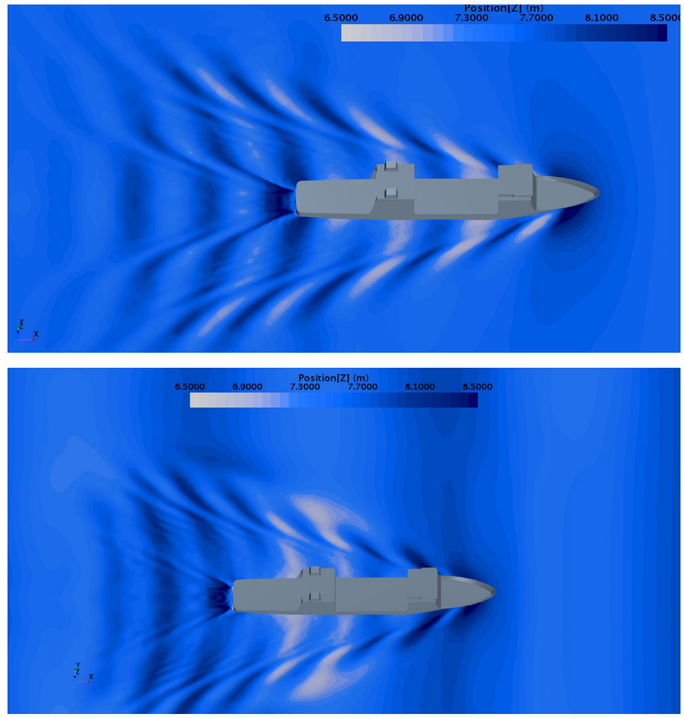

Figure 10: Pictured are predicted wave patterns around a full-scale ship in calm sea (upper) and with long-crested waves that are similar in length to ship length (190 m) with an amplitude of 0.5 m (lower). This example shows simulation of self-propulsion with a specified propeller rotation rate for the same vessel as figure 8.

Conclusion

In this paper the use of CFD simulation at full scale rather than model scale over a range of applications has been examined, with the aim of allaying some common reservations about using CFD at full scale in the marine industry. Analyzing wall boundary layer requirements and Reynolds number effects shows it is no more computationally expensive to perform simulations at full than at model scale, and the confidence in the accuracy of the results is no lower than the confidence in extrapolation of model experiments to full scale. In many cases, full-scale prediction is more reliable than scaling up model experiments, especially when these are carried out at subcritical Reynolds numbers.

Although full-scale measurement data for comparison and validation is limited, the results from the Lloyd's Register Workshop in November 2016 show the agreement between predicted speed/power curve from self-propulsion simulations and experimental data within 2 percent has been achieved by several groups. In another example, the prediction of wave-in-deck loads on a jack-up platform performed under full-scale conditions by Pakozdi et al. (2015)[4] agreed well with scaled up experimental data performed under model-scale conditions.

Many experienced users in the maritime sector are already routinely and successfully applying CFD simulations under full-scale conditions. For those who are still hesitating, it is time to start gathering experience since the trend is clear. Using Simcenter STAR-CCM+ makes it possible to meet the goal of conducting full-scale analysis of complete systems under realistic operating conditions by creating a digital twin of the real system.

References

1. Wosnik, M., Castillo, L., George, W.K.: "A theory for turbulent pipe and channel flows," Journal of Fluid Mechanics., vol. 421, pp. 115–145 (2000).
2. Lee, M., Moser, R.D.: "Direct numerical simulation of turbulent channel flow up to Re_τ 5200," Journal of Fluid Mechanics., vol. 774, pp. 395–415 (2015).
3. Lloyd's Register Workshop Proceedings link: http://info.lr.org/l/12702/2017-0220/3m372v/12702/156863/Proceedings.zip
4. Pákozdi, C., Östeman, A., Stansberg, C.T., Peric, M., Lu, H., and Baarholm, R.; "Estimation of wave-in-deck load using CFD validated against model test data," Paper No. ISOPE-I-15-586, ISOPE2015 Conference, Hawaii, 2015.

Siemens PLM Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Unit 901-902, 9/F
Tower B, Manulife Financial Centre
223-231 Wai Yip Street, Kwun Tong
Kowloon, Hong Kong
+852 2230 3333

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Digital Factory Division, is a leading global provider of software solutions to drive the digital transformation of industry, creating new opportunities for manufacturers to realize innovation. With headquarters in Plano, Texas, and over 140,000 customers worldwide, Siemens PLM Software works with companies of all sizes to transform the way ideas come to life, the way products are realized, and the way products and assets in operation are used and understood. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

www.siemens.com/plm

76061-A13 2/19 A