Efficient propeller Designs based on Full scale CFD simulations

N.W.H. Bulten, P.W. Stoltenkamp, J.J.A Van Hooijdonk

Wärtsilä Propulsion, R&D-Hydrodynamics, Lipsstraat 52, 5151 RP Drunen, The Netherlands

ABSTRACT

In order to be able to further enhance the performance of ship propellers, the possibilities of full scale numerical flow simulations have been investigated. The aim is to get a full understanding of the occurring flow phenomena on the actual ship. With this knowledge the optimum propeller design can be made. During the validation process a critical review of the model scale measurements methods has been made. The validity of some of the commonly used procedures has been evaluated. The use of full scale CFD simulations provide direct full scale data on the hull wake field and the propeller performance. It has been shown that the commonly used extrapolation methods predict different answers. Decomposition of the forces acting on the hull, the propeller and the rudder is being used to get a proper insight in the flow field at full scale. In the end the design features which contribute to efficiency increase, and thus fuel consumption reduction, can be isolated, based on the results from detailed flow simulations.

1. INTRODUCTION

Though the design process of fixed pitch and controllable pitch propellers are based on a long history, there is still a desire to further improve the efficiency of the propellers. In order to come to even better designs, it is important to evaluate the available knowledge and the research tools. The majority of the current knowledge on ship resistance, propeller performance and propeller-hull interaction has been derived from model scale measurements. During the last two decades the developments of numerical flow simulations have made such progress, that the use of Computational Fluid Dynamics (CFD) has become within reach in maritime industry. The required costs to get the simulations up and running and the achievable accuracy of the simulations are nowadays at a level that it can compete with experiments in model basins.

The insights obtained from numerical flow simulations have led to a close review of the methods applied in model testing. It is acknowledged that there have been made decisions in the past to follow a certain approach in model testing. Nevertheless, some constraints from experimental side, might not be present in numerical simulations, and therefore the approach can be revised in some cases. This point can only be reached after an extensive validation process of the numerical methods however.

In the following section the historical method for performance prediction based on model scale testing will be reviewed. This will give the background of the ideas where numerical flow simulations may have benefits. In section 3 the development of the numerical methods will be reviewed. Some results of the extensive validation process will be shown to give an impression of the achievable accuracy of the methods. Most validation work is based on model scale dimensions in order to be able to make a fair comparison with experimental data. Although full scale performance data is scarce, it is the authors opinion that the step to full scale numerical simulations should be made. Analysis of the differences in results from both model scale and full scale simulations can provide valuable information on the occurring Reynolds scaling effects. In such way an indirect evidence of the validity of the full scale results can be obtained.

Once the confidence in the full scale numerical flow simulations has been established, all kind of design variations can be analysed. Typical examples of cases studies can be propeller diameter variations to determine the effects on both open water efficiency and propeller-hull interaction losses, or implementation of energy saving devices like rudder bulbs or propeller boss cap fins. Results from some of these cases will be discussed in detail in section 4. The conclusions from the full scale numerical flow simulations will be drawn in the last section.

2. HISTORICAL PERFORMANCE DETERMINATION

For a very long time model scale experiments were the only way to collect data on the propulsive performance of the vessels. The extrapolation of the measured model scale data to the actual full scale performance of the vessel is partly based on an empirical approach. This process has been tuned by the different model basins over the years to come to quite accurate full scale performance predictions.

In order to get more insight in the overall vessel performance, there have been introduced three interaction factors. The background of these factors will be discussed in the following subsection.

2.1 Propeller-hull interaction factors

The coupling between hull resistance and propeller performance is based on the so-called interaction factors: wake fraction, thrust deduction, relative rotative efficiency. The conventional method of performance determination is based on a set of measurements:

- bare hull resistance
- propeller open water performance
- self-propulsion (combination of hull and propeller)

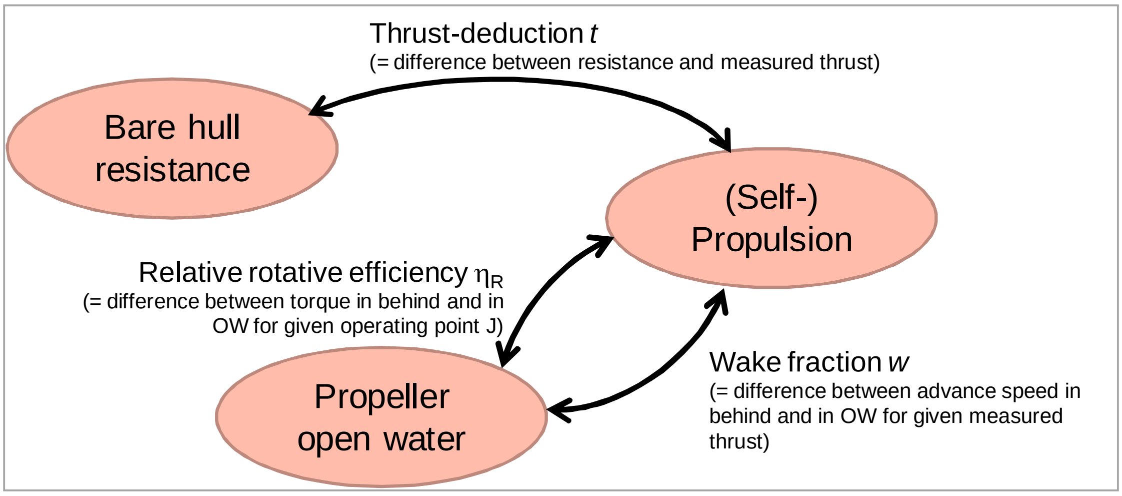

Fig. 1. Relations between model tests and interaction factors

It is known that the hull resistance in the self-propulsion test increases due to the working propeller, which creates a low pressure just upstream of the propeller. This increase in resistance is expressed as thrust-deduction factor t. The thrust-deduction couples the measured bare hull resistance with the measured thrust in the self-propulsion case. The inflow velocity to the propeller in behind the vessel is lower, due to the development of a boundary layer along the hull and the presence of shafts, skegs and brackets in front of the propeller. This velocity deficit is denoted with the wake fraction w. This wake fraction couples the measured thrust in the open water set-up with the measured thrust in the in behind condition. In order to get the bookkeeping closed there is one additional factor required, which gives the ratio between the power absorption in behind and in open water. This is the relative rotative efficiency η_R, which is for open propellers often close to 1.00.

2.2 Interpretation of interaction factors

Though the methods to calculate the interaction factors seems straight forward, significant differences in interaction factors can be found between different model basins. Given the method of data analysis and the impact of relative small deviations in the measurements, the interpretation of the interaction coefficients should be done with care. Nevertheless, over the years some rules of thumb have been derived to get rough indications of the interaction values. As a consequence the interaction factors have gained quite an important position in the performance prediction of ships.

With the introduction of integrated rudders, energy saving devices and ducted propellers, the use of interaction factors has continued, though this has led to non physical interpretation of interaction parameters in many cases. So, for more modern propulsion concepts and energy saving devices, the conventional approach of the interaction factors has been stretched over the limits of its validity. In those cases the possible gains in fuel consumption might end up in non physical interaction factors and as a consequence the phenomena might be discarded based on the motivation of measurement inaccuracies.

2.3 Review of open water performance test set-up

According to the commonly accepted procedures a propeller is driven from the downstream side in an open water performance test. In this way the inflow to the propeller is as uniform as possible. In case the propeller has to be driven from upstream side, the drive mechanism will create a flow disturbance which is undesirable. The choice to put the drive shaft downstream has had the consequence that the impact of the hub cap design has not been taken into account in the propeller performance measurement for decades. This is shown in fig. 2.

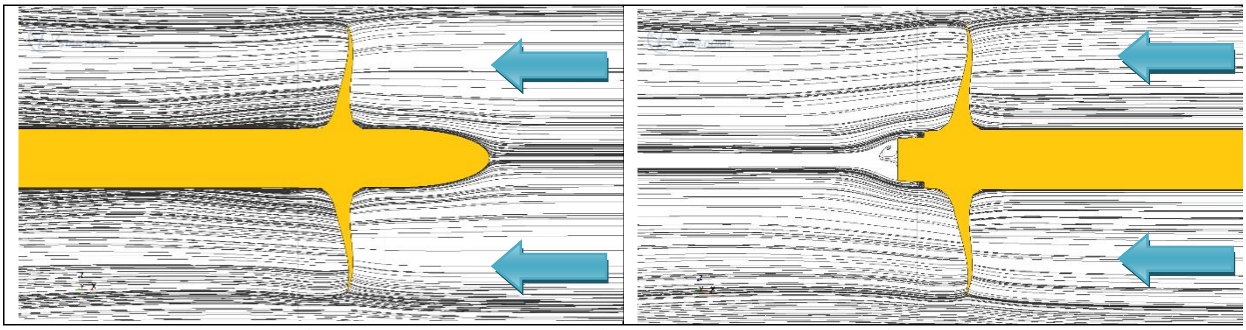

Fig. 2. Comparison of open water test set-up configuration and set-up in self-propulsion in behind ship

In the self-propulsion measurements, where the actual hub cap shape is modelled properly, possible gains or losses are part of the overall performance measurement. Any possible impact of the performance of the hub cap will therefore end up in one or more of the interaction or correction factors. The unavoidable inconsistency in the open water testing method will not reveal the true performance of the applied hub cap shape. The flow simulations for the example case as shown in fig. 2 revealed an efficiency drop of about 2%. In the conventional model testing approach, these effects will end up in the various interaction factors, and become untraceable in that way. Moreover the development of energy saving devices like propeller boss cap fins has suffered from this test methodology. With numerical flow simulations the potential benefits of boss cap fins can be determined properly, both on model scale and full scale (Kawamura 2013).

3. DEVELOPMENT OF NUMERICAL METHODS

In the following subsection the current status of the viscous flow simulations will be described. This should give a good impression of what can be expected as state-of-art methodologies nowadays. An important aspect of the implementation is the validation of the numerical methods. This will be discussed in the subsequent subsection, where some results of the extensive implementation and validation process will be discussed. In subsection 3.3 an evaluation will be made of the added value of full scale simulations. Typical phenomena which are occurring in model scale testing might be avoided with aid of numerical simulations. Examples are among others the Reynolds scaling effect on wake fields and ducted propeller performance and the sensitivity analysis of hull roughness on resistance and performance.

3.1 Current status of numerical simulations

The development of numerical methods is a continuous process. Significant steps forward are being made both on the hardware side and on the commercial software side. In general these developments are globally driven. It is therefore expected that the developments will continue further in the coming years. Based on this assumption, it is worthwhile to start working on method development of numerical flow simulations, which might at this point in time not yet be suitable for daily commercial use. However, by the time the methodologies have reached a certain maturity level, which means sufficiently validated, and captured well in process descriptions and procedures, the lead times and costs have reduced enough for commercial applications.

Nowadays the effects of viscous flow can be taken into account, which means that accurate bare hull resistance predictions are feasible. Based on current technology the viscous flow simulations (also denoted as RANS (Reynolds-Averaged Navier-Stokes)) can take the effects of the free surface (VOF) and the dynamic sinkage and trim into account. And the accuracy of the calculations can compete with the accuracy of the model scale measurements.

Moreover, there will be differences in the results when model scale and full scale geometries are compared. Due to the proper calculation of the full scale viscous flow effects the need for the semi-empirical extrapolation methods, as used in the model tests, will diminish.

The following step in the development of the numerical simulations is the propulsion calculation, where the ship and the propeller are analyzed together. The propeller performance is then derived from fully transient moving mesh simulations with sliding interfaces. In these simulations, the propeller position is adjusted every time step, which gives the time dependent solution of the flow. The propeller thrust and torque are calculated for each time step in this approach.

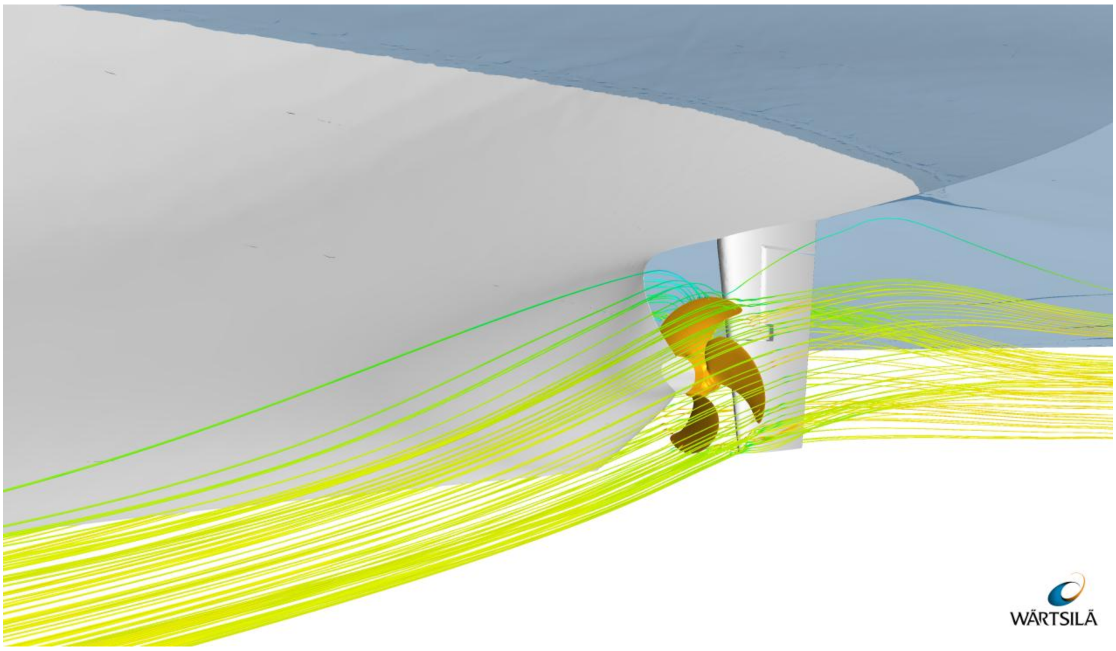

Fig. 3. Numerical flow simulation of hull and propeller, including free surface effects

The added value of the numerical simulations is found in the extensive options of flow visualization (see for example fig. 3) and post-processing. With these other means of data analysis it is possible to get new insights on the actual occurring flow phenomena, like the interaction phenomena. It is also possible to determine the drag contribution of different components and appendages on the hull to get an indication of the contribution to the total resistance.

3.2 Implementation and validation of numerical methods

Numerical flow simulations in maritime industry have gained maturity during the last years. One of the important issues in this process is the quality assurance of the simulations. This covers not only the level of achieved accuracy, but also the processes and procedures to reach the repeatability of the results. The target should be that proper implemented numerical methodologies are independent of the expert who is carrying out the simulations. The methodology should also be robust enough to handle different ship and propeller designs. This target can be achieved with parametric mesh topology definitions in general.

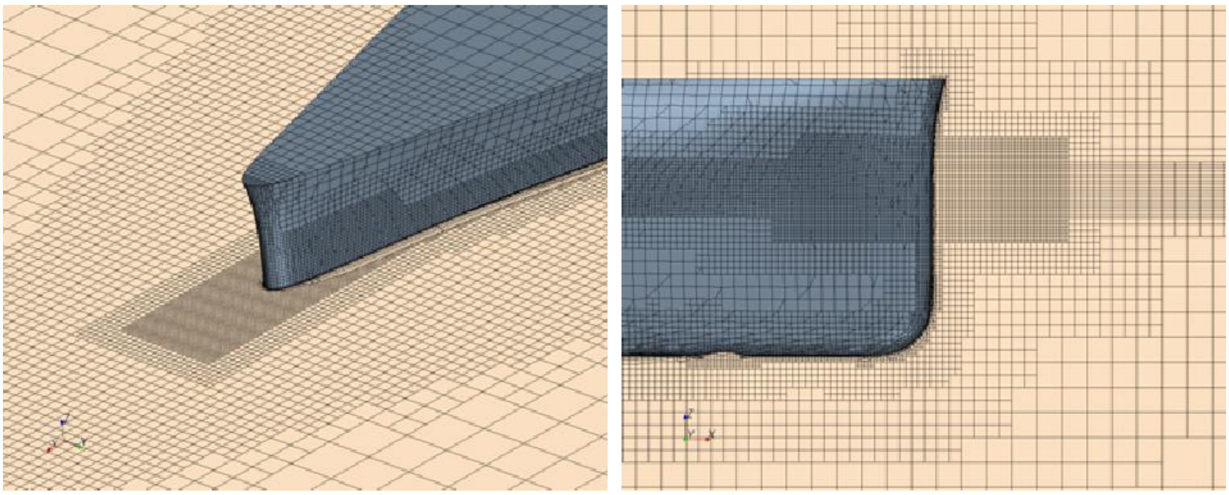

Fig. 4. Mesh near bow with local refinements to capture free surface effects

An example of the mesh topology around the bow of the hull is shown in fig. 4. In order to get a good balance between total cell count and required mesh density, local mesh refinement near the free surface is implemented. The actual free surface is based on the solution of the Volume of Fraction (VOF) of the water and air mixture. The final meshing approach has been the deliverable of the implementation and validation process.

The outcome of a bare hull resistance calculation with free surface is shown in fig. 5, where the model scale resistance is shown for different vessel speeds. The agreement between the measured and the calculated values is good over the complete range of ship speeds. Similar results have been found for other vessels, which indicate the robustness of the method.

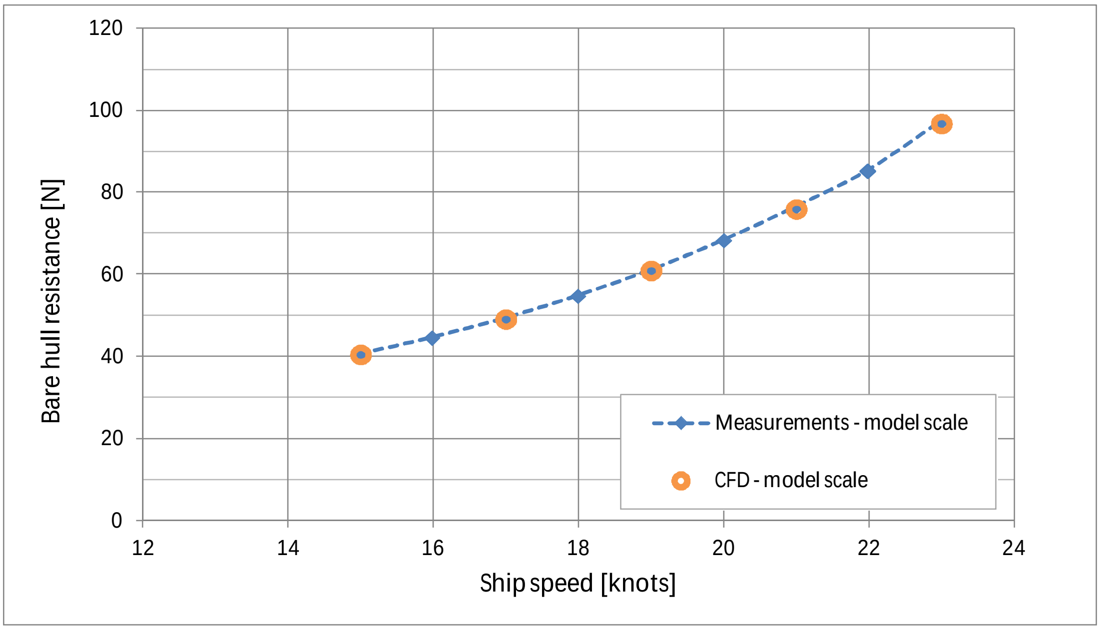

Fig. 5. Comparison of bare hull resistance measurements and calculations

Besides an accurate calculation of the bare hull resistance it is also required to implement a proper propeller performance methodology. The quality of such method can be verified with open water model scale performance measurements. In the marine industry, the critical success factors for propeller performance calculations are regarded to be the ability to model the blade geometry with sufficient detail (meshing), proper implementation of the rotation of the propeller and selection of the turbulence model. Fig. 6 shows the open water performance curves for a selected open propeller. For this propeller two well-known turbulence models, k-ε and k-ω-SST, have been used to calculate the open water performance on model scale. The figure shows that comparable results can be obtained with both turbulence models. At low advance ratios a slightly different trend can be observed, which may be attributed to a small pitch effect due to differences in boundary layer development along the blade. However, the overall performance comparison indicates that both models can be used in the numerical simulations. Comparison of the numerical results with the model scale measurements learns that the overall trends of thrust and torque are captured well over the whole range of advance speeds. Though there is a difference between the measurements and the calculations near the top efficiency. This is a result of the fairly subtle differences in the thrust and torque values, which might be a result of some laminar flow effects or not properly reported hub cap correction factors in the experiments in the end. Near the design point of the propeller, which is around J=0.65, the agreement between measurements and calculations is better.

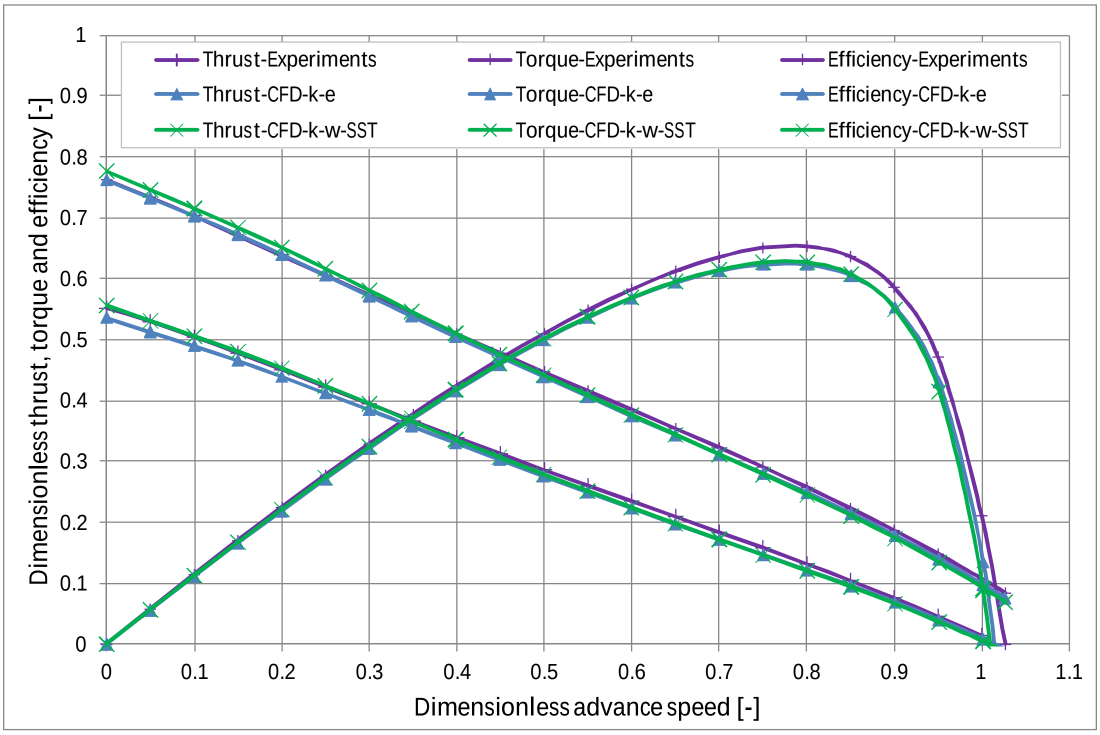

Fig. 6. Comparison of open water performance curves for open propeller based on model scale measurements and numerical simulations with k-ε and k-ω-SST turbulence models

Implementation of the propeller rotation in the numerical simulations can be either based on the quasi-steady Multiple-Frame-of-Reference (MFR) approach or the full transient moving mesh (MM) approach. In the MFR method the mesh remains fixed in a frozen position and the additional terms due to centrifugal and Coriolis forces are implemented in a spin domain. In this approach the steady solvers can be used leading to relatively short calculation times. In case the fully transient moving mesh option is used, a specified domain around the propeller is rotated every time step, resulting in a fully transient solution of the flow. The MFR approach works well for propeller open water calculations, where a uniform inflow is present and no flow disturbances are present behind the propeller. Apart from the propeller blade geometry an axi-symmetrical flow problem is being solved.
For propulsion calculations initially the MFR method has been applied. This gave results which looked plausible, but were wrong. The concept of averaging several MFR calculations, with different blade positions, is not recommended by the authors either. Therefore the fully transient moving mesh approach is regarded to be the only valid alternative for propulsion calculations, though it will be most computational intensive.

3.3 Added value of full scale numerical simulations

Even though there are still differences between model scale measurements and numerical simulations, it is interesting to investigate further steps on the side of the simulations. Calculations made for the actual full scale can provide knowledge and understanding of the occurring Reynolds scaling effects. It is also known for long time that some typical phenomena suffer from the Reynolds scaling effects, like the friction along the hull (ITTC 1957 friction line) and the propeller performance effects (ITTC 1978 ΔKt and ΔKq), which are addressed in several ITTC-conferences (ITTC 2011). Also the differences between model scale and full scale wake fields and the consequences for the propeller loading have been acknowledged (Ligtelijn 2004).

Another issue from model testing, which can be investigated easily with numerical simulations, is the location of the drive shaft, either upstream or downstream. In the simulations the drive shaft can be modelled in various ways, since it has no actual functionality. In this way a sensitivity analysis can be carried out. Such sensitivity analyses may also help in understanding the working principles of various energy saving devices, like propeller boss cap fins and rudder bulbs.

The use of full scale propeller performance data has an impact on the selection of optimum propeller diameter. The common approach is to use B-series data, derived from the published polynomials (Oosterveld 1974). With a performance polynomial of full scale B-series, a comparison can be made on the selection of the optimum diameter, based on model and full scale performance. Once this point has been reached, the true value of the capabilities of the numerical simulations comes to the surface.

The added value of the simulations do not limit to the design phase of the vessel and propeller. Once the vessel is in service, it is of interest to get a good indication of the effects of increasing roughness of the hull and the propeller on the fuel consumption. At certain point in time there will be a trade off between the cost of cleaning and the gains in fuel costs during operation. Numerical simulations with various grades of hull roughness can be of help in such evaluation.

4. RESULTS FROM FULL SCALE CFD PERFORMANCE SIMULATIONS

In this section the aforementioned topics will be analysed in more detail, based on the results from the flow simulations. First the wake scaling effects will be shown and afterwards the effects of Reynolds scaling effects on propeller open water performance and the impact on the propeller diameter selection. Finally, results from the work on energy saving devices will be shown.

4.1 Wake scaling

The wake field of a vessel is one of the key input parameters for the propeller design process. Once the bare hull design is available and resistance measurements are carried out, often the wake field is measured as well. It has been recognized that the model scale wake field will differ from the actual full scale wake field (Benedek 1968), due to Reynolds scaling effects. The boundary layer development along the hull causes not only a difference in hull friction, but also in the velocity distribution at the aft part of the vessel, where the propeller will be located. For the propeller designer two factors in the wake field are of importance in the propeller design process: the depth of the wake peek in the top position and the gradients of the velocity along a circular path. These parameters determine how and to which extent the blade load fluctuates during a revolution. A comparison of two calculated wake fields is shown in fig. 7 for a single screw vessel. The wake field, as calculated for model scale, shows a deeper wake peek compared to the full scale wake field. Moreover, the overall wake fraction w will be larger for model scale compared to full scale.

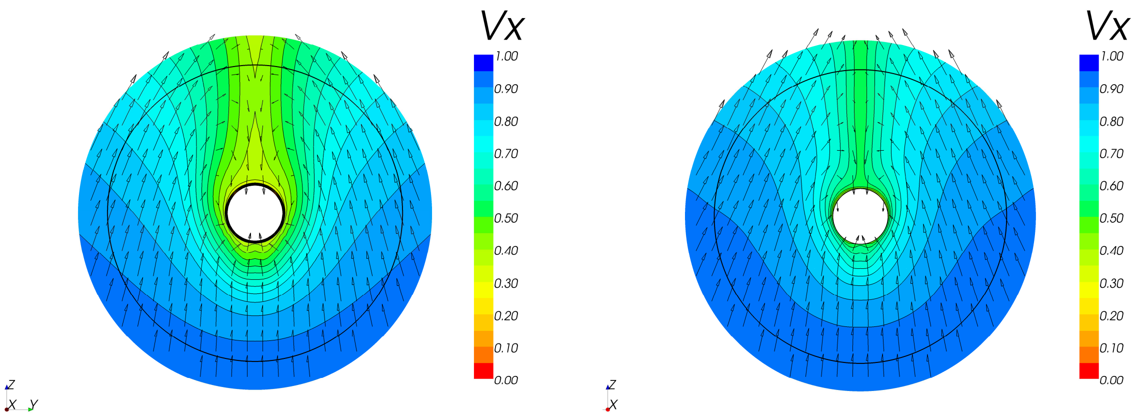

Fig. 7. Comparison of calculated wake field on model scale (left) and full scale

4.2 Propeller open water performance

For the scaling of propeller open water performance ITTC has made a procedure back in 1978 (ITTC 1978). This scaling method introduces an offset on the non-dimensional thrust and torque values Kt and Kq. Due to reduction of blade friction, it is expected that the thrust increases slightly and the torque decreases at the same time. More recent CFD studies have shown that such a trend could be observed for certain families of propeller designs. However, there were also indications that for other families of propellers, for example with more skew, other phenomena played a role. For skewed propellers, there seems to be a pitch effect on the blades, which results in a shift of both thrust and torque in the same direction (Minguito 2005).

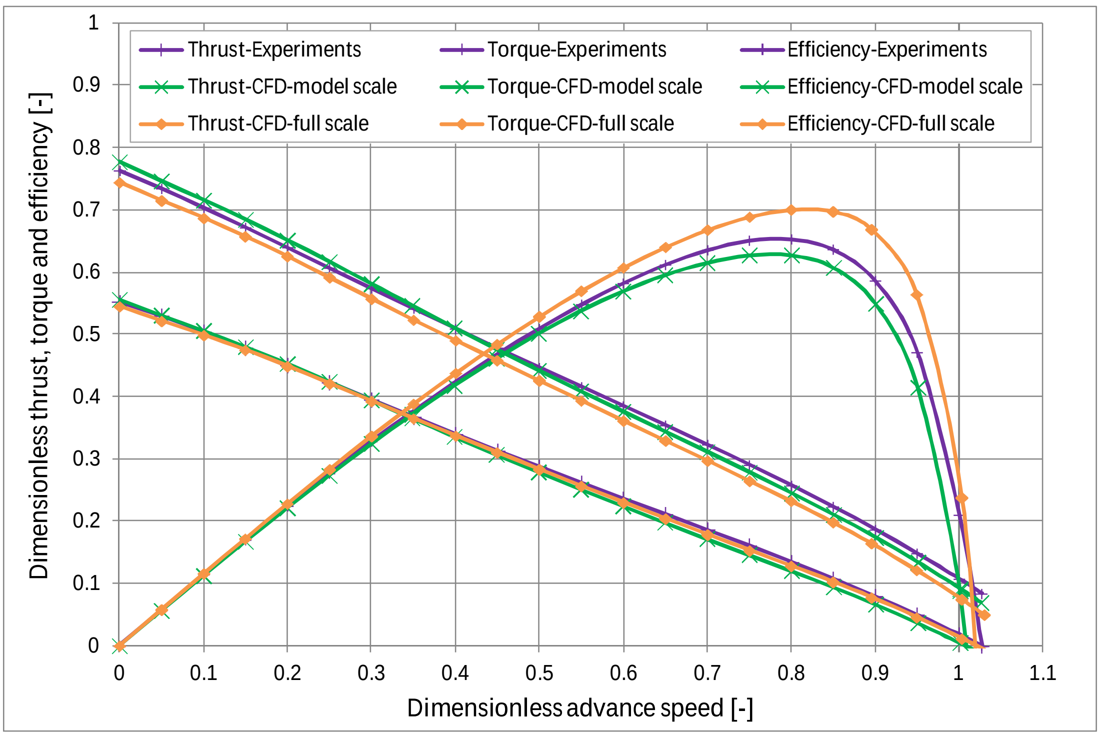

Fig. 8. Open water performance for calculated model scale and full scale and measured data

The Reynolds scaling effect on propeller performance is presented in fig. 8, where the calculated open water curves for model scale and full scale are shown, together with the experimental data. This diagram shows a quite significant difference between the model scale and full scale calculations. The torque at full scale is reduced and the thrust is increased slightly, which is in line with the expectations. The averaged ΔKt and ΔKq have been derived from the CFD results and compared with the values from the ITTC'78 method. The values are shown in table 1. Significant differences are found between the two methods. As a consequence the full scale performance prediction based on the ITTC'78 method will differ significantly from the calculated full scale curves.

Table 1. Comparison of ΔKt and ΔKq

	Δ Kt	10Δ Kq
ITTC'78	0.0003	-0.0029
CFD	0.0015	-0.0194

The differences in open water performance, due to Reynolds scaling effects, do have an impact on the selection of the optimum diameter for a given application. Based on the overall propeller powering characteristics, like power, RPM, ship speed, an optimum propeller diameter can be selected. This process is often based on the available Wageningen B-series polynomials, which have been derived from the model scale experimental data.

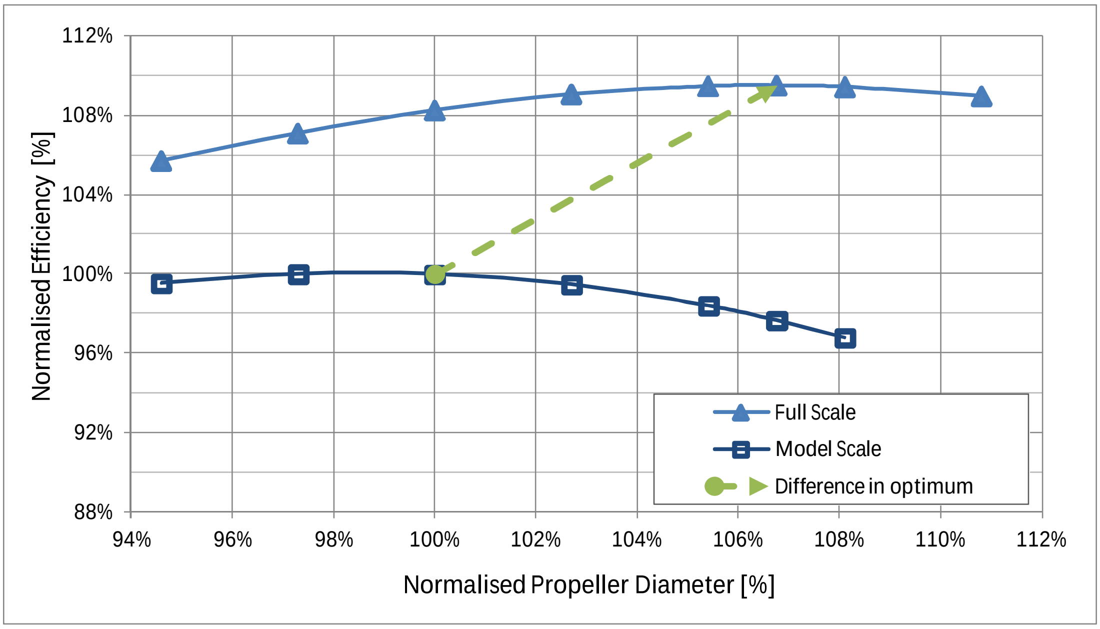

Fig. 9. Comparison of propeller diameter selection based on B-series model scale and full scale

Recently, a similar polynomial has been generated for full scale geometries based on CFD. With these results it has become possible to carry out the propeller diameter selection process based on both model scale and full scale data. The results are shown in fig. 9 and the outcome is quite remarkable. Based on the full scale open water data a 7% larger propeller diameter would have been selected.

4.3 Energy saving devices

The energy saving devices (ESD), like propeller boss cap fins and rudder bulbs, have attracted quite some attention in the last years. Though these concepts differ a lot in geometry, they have in common that the flow near the propeller hub is influenced mostly by these devices. In order to be able to give a proper evaluation of the effectiveness of these ESDs, it is important to understand the occurring flow phenomena near the hub and to minimize the impact of laminar flow and Reynolds scaling effects. Full scale numerical flow simulations of the hull, propeller and rudder can potentially provide the proper information to reach this goal.

CFD calculations of a vessel with and without rudder bulb have been made to investigate the differences in performance (see fig. 10). The major part of the hull resistance will not change and therefore differences in overall efficiency of a few percent are found. Nevertheless the components which are responsible for the performance gains can be isolated. Based on this analysis the coupling between efficiency gains, occurring flow phenomena and actual physical principles can be made.

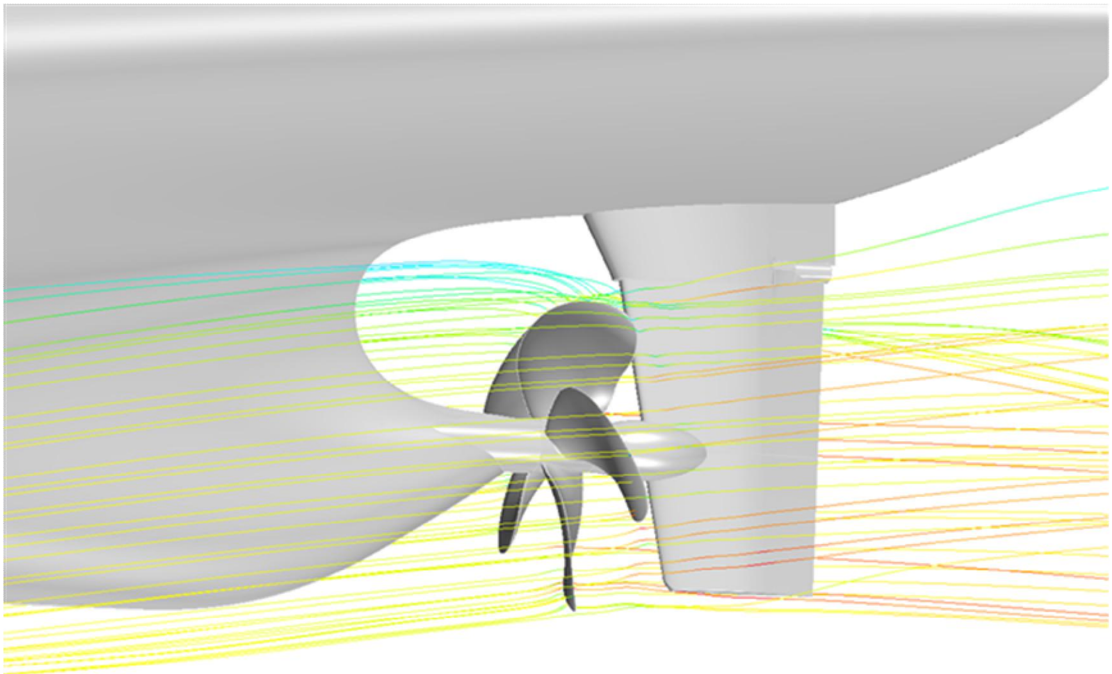

Fig. 10. Numerical flow simulation of hull, propeller and rudder with rudder bulb

5. INCORPORATION IN DESIGN PROCESS

In this paper various aspects of both model scale testing and full scale numerical flow simulations have been discussed. Though there are still steps to be made on the development of numerical simulations, the added value can be utilized in the design process of propellers. Fig. 11 shows a process description of a propeller design process. The conventional process, based on model scale experimental data, is described in the top half. In the lower half the possibilities of full scale numerical flow simulations are incorporated. On the input side of the propeller design process a full scale wake field and propeller diameter can be used. Once the design is available, either open water and self-propulsion performance evaluations can be made based on full scale simulations or on the conventional model scale measurements.

6. CONCLUSIONS

In order to be able to further enhance the performance of ship propellers it is necessary to have a full understanding of the occurring flow phenomena on the actual ship. In the past the majority of research and development in this field was based on model scale experiments. Nowadays it has become possible to get more detailed insights in the flow field when numerical flow simulations are carried out. Proper understanding of the propeller open water efficiency and propeller-hull interaction factors can be achieved with CFD simulations.

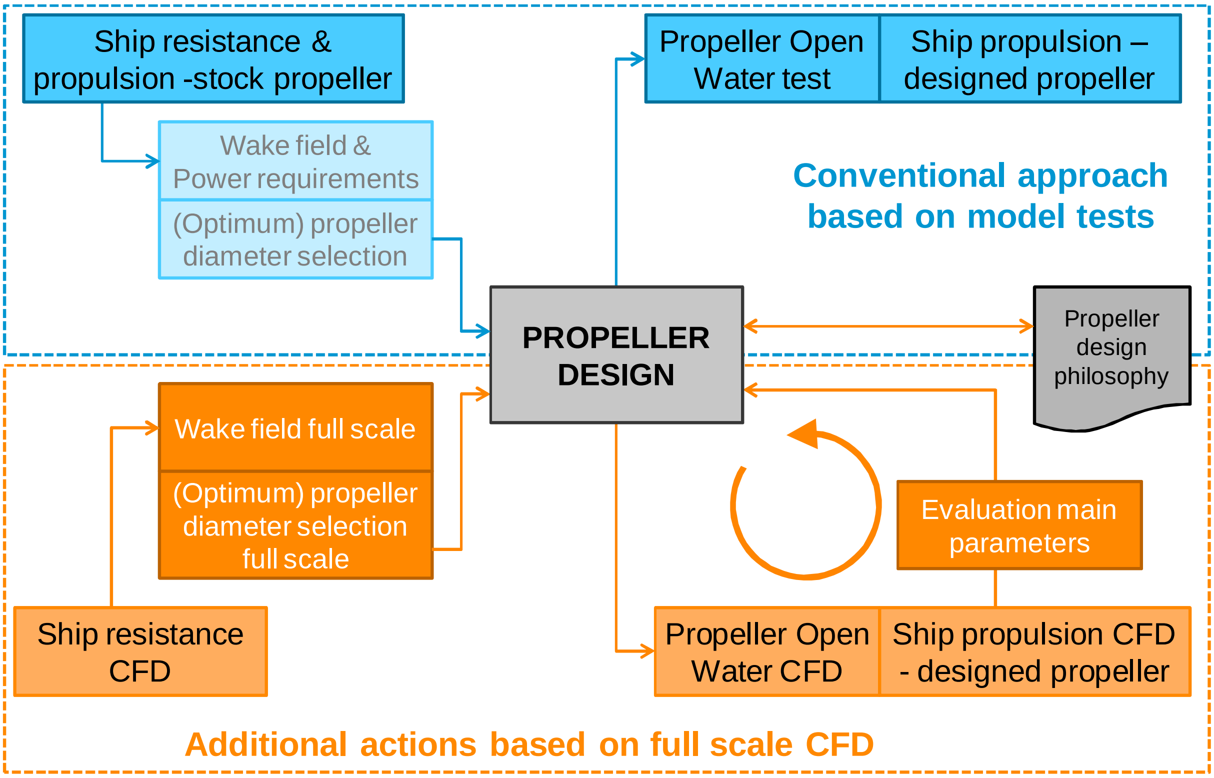

Fig. 11. Flow chart of implementation of full scale flow simulations in design process

Whilst gaining more knowledge and understanding on the flow effects, also some commonly used procedures in model scale testing can be revised. In the end the design features which contribute to efficiency increase, and thus fuel consumption reduction, can be isolated and further improved, based on the results from detailed flow simulations.

ACKNOWLEDGEMENTS
The authors would like to thank the CFD-experts from Wärtsilä Business Line Propulsion for distributing the results from their simulations.

References

Benedek, Z and Balogh B., "The scale effect on nominal wake fraction of single-screw ships", Periodica Polytechnica, Vol 12, No 1, 1968

ITTC, "1978 ITTC performance prediction method", Section 7.5-02-03-01.4,1978

ITTC, proceedings of 26th ITTC, Rio de Janeiro, 2011

Kawamura, T., Ouchi, K., Takeuchi, S., "Model and full scale CFD analysis of propeller boss cap fins (PBCF)", Proc. of Third International Symposium on Marine Propulsors, pp 486- Launceston, Australia, 2013

Ligtelijn, J.T., Van Wijngaarden, H.C.J., Moulijn, J.C., Verkuyl, J.B., "Correlation of Cavitation: Comparison of Full-Scale Data with Results of Model Tests and Computations", SNAME Maritime Technology conference, Washington, USA, 2004

Minguito, E., "Assessment of scale effects on propeller performance", Dissimination workshop EU-project Leading Edge, Copenhagen, Denmark, 2005

Oosterveld, M.W.C. and Van Oosanen, P., "Further Computer-Analysed Data of the Wageningen B-screw Series", IV International Symposium on Ship Automation, Genova, Italy, 1974