Fifth International Symposium on Marine Propulsion
SMP'17, Espoo, Finland, June 2017

Full scale CFD: the end of the Froude-Reynolds battle

Norbert Bulten, Petra Stoltenkamp

Propulsion Technology - Hydrodynamics,
Wärtsilä Netherlands, Drunen, The Netherlands

ABSTRACT

The performance of various propulsion products has been determined with numerical flow simulations. The results of model scale and full scale CFD simulations have been compared and clear trends on the observed Reynolds scaling effects have been identified. In this paper the results from research from the last decade have been combined into a clear message that full scale numerical flow simulations are to be used in the (near) future. The arguments for model testing are put in a new perspective and they should be balanced against the arguments against usage of full scale CFD simulations.

The presence of laminar flow effects in model scale experiments might occur more often than anticipated and the impact should be acknowledged. The commonly applied extrapolation methodology (ITTC'78) estimates a performance gain of a few percent, which seems to be valid for conventional open propellers only. All kind of other, more modern, propulsion configurations, like ducted propellers and thrusters, show significantly larger Reynolds scaling effects, due to the turbulent flow regime on both model and full scale.

Keywords
RANS, CFD, extrapolation, cavitation.

1 INTRODUCTION

For more than a century research and development in marine industry has been based on model scale testing, as a result of the application of Froude's law. This approach has worked well for a long time, but it has always been difficult to take the effects of the Reynolds number into account. So for a long time the battle between Froude and Reynolds was in favor of the first.

With the development of viscous numerical flow simulations (CFD) it has become possible to analyze the flow at actual ship scale, which means both Froude and Reynolds number will be equal to the actual situation. One of the critical points in CFD remains however the validation of the numerical results. A common approach is to validate the methodology first on model scale and then make the proper adjustments to handle the full scale Reynolds number flow. Recently, a data set of full scale measurements has been made available by Lloyds, which has opened the route to direct full scale comparisons. With the results from full scale CFD simulations it has become possible to review the existing extrapolations methods based on ITTC'78 (Ponkratov & Zegos, 2015). This methodology has been developed to predict the full scale performance of vessels with conventional open propellers. Nowadays, tip-plate propellers, energy saving devices, like pre-swirl stators, nozzles and rudder bulbs among others are common features. It should be noted that the actual impact of these devices might not be predicted correctly with the conventional extrapolation methods (Sánchez-Caja *et al*, 2014). Moreover, it is to be expected that the method contains several 'errors', which cancel each other out to a great extent. As shown by Guiard *et al* (2013). A comparison of a wakefield on model scale and full scale is shown, in combination with the impact of hull roughness. From this analysis it can be seen that the additional friction due to hull roughness leads to a similar wakefield as found on model scale, where the friction is increased due to the lower Reynolds number.

In case of ducted propellers the scaling from model scale to full scale is also a combination of two effects (Bulten&Nijland, 2011): (i) the reduction of friction results in a reduction of torque and an increase of thrust, (ii) due to the increased flow rate through the nozzle, both the propeller thrust and torque decrease. Since both effects are of similar magnitude, the final impact on propeller torque can be positive as well as negative.

The examples mentioned above clearly indicate that the actual occurring flow phenomena can be explained well, once it has been fully understood. It also indicates that the conventional approach of model scale experiments and extrapolation methods has reached its limits. It is therefore useful to review the merits of the alternative route, as shown in Figure 1, where the answers are derived from full scale CFD simulations.

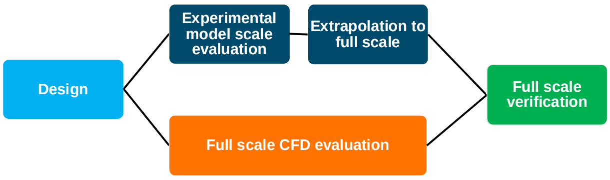

Figure 1: design-evaluation-verification chain

In this paper two topics will be addressed related to the scaling effects. First a detailed analysis will be presented on the propeller open water performance predictions. Second topic is related to the observed cavitation of a pushing azimuth thruster unit, where the propeller operates in the wake of the strut.

Another point of attention, outside the scope of this paper, is the selection of the actual design point. Often the trail condition is a once in a lifetime sailing condition, for which a large part of the machinery is being designed and optimized. New verification methods might have to be developed in order to judge the actual performance on more realistic operating conditions.

2 PROPULSION CONFIGURATIONS

2.1 SHAFTLINE DRIVEN UNITS

Performance calculations have been carried out for a large variety of propulsion configurations, from open propellers to azimuth pushing ducted propellers. In the open water analyses of the open propellers, both fixed pitch (FPP) and controllable pitch propellers (CPP) have been analyzed. The open water performance determination is identical for both types of propellers, but still it is expected that the actual behavior might differ when both types are compared. The distribution of the chord-length from hub to the tip is often quite different. The controllable pitch propeller blades need to be able to pass each other when the pitch is set to negative. This is a clear limit on the allowable chord length. Moreover the available chord-length near the blade foot is limited by the size of the blade foots in general. As a consequence of the various limitations, still a fairly large chord length near the 0.7 radius is observed for many CPP designs. For FPP designs, the chord lengths and blade area ratios (BAR) can vary between 0.40 and 1.00. Where the lower BAR is representative for propellers for tankers and bulk carriers and the higher values for the more conventional container vessels with a design point above 25 knots.

For applications up to 15 knots it can be beneficial to add a nozzle (or duct) to the propeller. Some well-known duct types are the Marin 19A and 37 nozzles, where the 19A has a relative sharp trailing edge and the 37 a rounded shape, which allows sailing astern. Over the years Wärtsilä has developed some variants, denoted with HR and HPN for the shaftline driven propellers and the WTN series for the azimuth thrusters. One of the clear differences is the shape of the trailing edge, where a much sharper trailing edge is applied for the thruster nozzles. In case of astern thrust, the unit can be rotated easily with 180°. Nozzles, which are optimized for bollard pull condition, are executed with a diffuser shape to enlarge the amount of flow through the unit. On the other hand, applications for 12-15 knots transit speed, work best with shorter nozzles with a cylindrical inner shape.

For optimum performance of a ducted propeller, the clearance between the propeller blade tip and the innerside of the nozzle should be as small as possible. For FPP-designs it is possible to apply a significant tip length of the blades with small clearance. For the CPP-designs, the requirement for pitch deflection, limits the maximum blade tip length, to avoid collision of the propeller with the nozzle.

Based on the theoretical approach of the performance of ducted propellers from 2011 (Bulten & Nijland, 2011), it can be concluded that the extra tip length of an FPP results in a higher pumping efficiency and therefore in a higher overall performance.

2.2 AZIMUTH AND TUNNEL THRUSTERS

The open water performance of azimuth thrusters has been determined in aligned flow, representing 0° steering angle. Thruster units can be characterized by three main items: (i) pushing / pulling, (ii) open / ducted, (iii) FPP / CPP. An analysis of the occurring scaling effects has been presented in (Bulten & Stoltenkamp, 2017) for both a pulling thruster with open controllable pitch propeller and a pushing thruster with ducted fixed pitch propeller.

Since a number of years the tilted gearbox configurations have been introduced for azimuth thrusters, to reduce the thruster-hull interaction losses. These thrusters have an 82° gear transmission and a propeller shaft line with 8° downward tilt. Due to this tilt and the shape of the nozzle, the jet out of the thruster is deflected sufficiently downwards to limit the negative interaction with the hull surface.

Tunnel thruster performance calculations can be made in an isolated environment, where the unit is placed in an infinite tunnel or in an actual hull geometry. In case the tunnel thruster is analyzed in the infinite tunnel, the performance can be determined for different flow rates, or advance speeds, similar to a propeller open water test. When the complete hull geometry is taken into account, the flow rate through the system becomes a part of the solution. The concept of the minimal tip-clearance gap, as described for the ducted propeller is applicable to tunnel thrusters as well. The performance of an FPP-design with tip length is therefore significantly better compared to the CPP alternative.

3 CFD METHODOLOGY

3.1 MODELLING APPROACH

The equations of mean flow motion are calculated using standard flow solving techniques: a quasi-steady segregated flow solver is selected to resolve the flow field. The numerical schemes are based on the SIMPLE algorithm (solves pressure and velocity fields separately). Hybrid Gauss-LSQ technique is used to calculate the gradients. A moving frame of reference (MFR) takes the propeller rotation into account. The equations of turbulent motions are modelled using the RANS k-ω SST (Menter) turbulence model.

Default Star-CCM+ settings of the k-ω SST model remain unaltered. For the inlet boundary condition a uniform velocity profile is prescribed. Pressure conditions are applied on the outlet and circumferential boundaries of the cylindrical domain. Wall-bounded boundaries obey a no-slip condition. For the open water performance calculations, gravity effects were considered to be irrelevant and are therefore excluded. Simulations which include the cavitation model are carried out with gravity included, because the impact can be considerable at actual full scale.

An open water simulation is solved using a two-stage approach: first order discretization schemes are selected for the first 100-200 iterations. The additional 800-1200 iterations are then solved using second order spatial schemes. The total number of iterations varies, depending on several convergence criteria. Apart from monitoring the transport residuals, the main convergence criteria are based on minimal variation levels of thrust, torque and mass flow through the nozzle. Flow is considered to be converged when the asymptotic variations become smaller than 1% of their sliding sample window mean values (last 500 iterations). For transient simulations including cavitation a more complex solution strategy is applied to get the optimum balance between overall calculation time and achieved resolution in time, as described in more detail by Bijlard & Bulten (2015)

3.2 MESH SPECIFICATION

Unstructured hexahedral cells are used to build the computational grid. The computational domain is divided into two regions: a non-rotating main cylinder, and a rotating MFR domain. The propeller, part of the shaft, the hub/cap, and nozzle are placed in the rotating domain. The upstream distance between the inlet and the propeller generator line is set to 5Dprop. Grid refinements are applied to important feature curves of the 3D CAD geometry, as shown in Figure 2 for the ducted propeller case. A fine volume grid refinement is applied to the direct surrounding of the propeller and inner region of the nozzle. An intermediate volume grid refinement covers a slightly larger area than the MFR-domain. Edge refinements were applied to leading and trailing edges of the propeller. The trailing edge of the nozzle got a similar treatment together with an extended wake refinement. Furthermore, in order to obtain sufficiently smooth flow statistics for propeller and nozzle surfaces a maximum surface size discretization was set. The computational grid near solid walls are all defined according to a standard geometric stretching prism layer distribution. The thickness of the near-wall prism layer is typically set to 5µm on model scale, but could be manually adjusted to obtain smaller y^+ values. The total thicknesses of the prism layers are set to 5mm. In total 10-15 layers are incorporated. The quality of the near-wall grid refinement was evaluated by checking the y^+ values for all simulations. Virtually all y^+ values are located in the viscous sublayer to ensure proper near-wall resolution

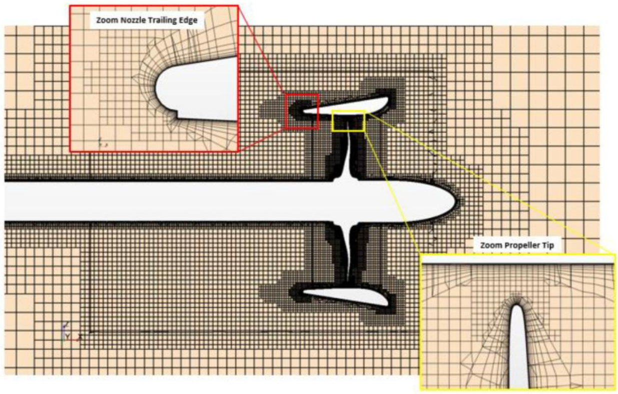

Figure 2: mesh example of ducted propeller

4 OPEN WATER PERFORMANCE

In this section the results of the CFD simulations for the various propulsion units will be discussed. The CFD methods have been validated in earlier stage and for the different type of units, these validated methods have been applied without further tuning or adjustments.

For the open propellers, a population of about 40 propellers has been analyzed, which has become within reach, due to the highly automated process of open water performance calculations. For the ducted propellers, quite a set of units has been analyzed. Only a limited number of results will be presented in this paper, due to confidentiality reasons. The thruster simulations are limited to a small group of geometries, due to the large similarities in the thruster geometries of the Wärtsilä Steerable Thruster-series (WST).

4.1 OPEN PROPELLERS

A large population of FPP and CPP propellers have been analyzed on model scale and full scale. It has been identified that for a number of propellers an excellent match of the open water curve could be obtained, when a laminar flow regime was used in the CFD simulations (Bulten, 2015). In order to get a better understanding of the occurring phenomena, it has been decided to run the open water simulations in both laminar flow regime and in fully turbulent flow regime.

As shown in Figure 3, a very good match is found for a number of propellers, based on the laminar regime. Propeller thrust, torque and efficiency show all a good agreement with experimental data over the whole range of J-values. On the other hand, a number of propellers have been identified, where the match of the measured thrust, torque and efficiency with the fully turbulent CFD simulations with k-ω SST model is observed. Korkut and Atlar (2002) indicated that model scale tests could suffer from a lack of free-stream turbulence, which had a clear influence on the cavitation inception behavior of the propellers in their research. This lack of free stream turbulence can thus be linked to the observed laminar performance behavior as well.

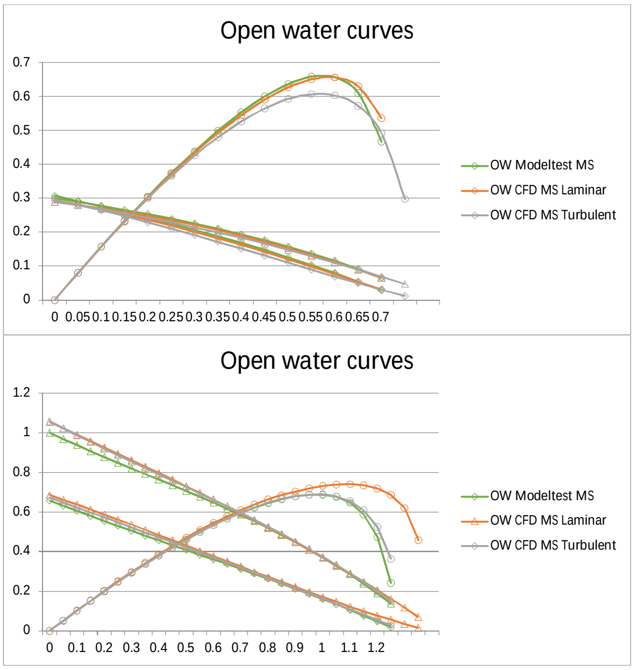

Figure 3: open water comparisons of CFD results with model scale measurements based on laminar flow regime (top) and turbulent flow regime

Based on the findings for the large population of open propellers, a more detailed analysis has been made, with the aim to find a method to make a simple distinction between expected laminar or turbulent behavior. For this analysis an InterpolationFactor has been defined, which calculates the amount of turbulence based on the measured open water efficiency in the design point and the calculated results for laminar and the turbulent flow regime:

$$\text{InterpF} = \frac{\left(\eta_{0-\text{CFD-lam}} - \eta_{0-\text{EXP}}\right)}{\left(\eta_{0-\text{CFD-lam}} - \eta_{0-\text{CFD-turb}}\right)} \quad (1)$$

This simple factor gives a result of 0% in case the match with the laminar flow regime is found and 100% in case the measurement matches the turbulent flow regime.

In Figure 4, the InterpolationFactor is plotted against the local Reynolds number of the chord length at 0.7Radius ($C_{0.7}$), where the $Re_{c-0.7}$ is defined as:

$$Re_{c-0.7} = \frac{\left(\pi \times n \times 0.7 \times D \times C_{0.7}\right)}{\nu} \quad (2)$$

where n is the propeller rotational speed, D the propeller diameter and ν the viscosity.

In this diagram the different markers represent different model basins.

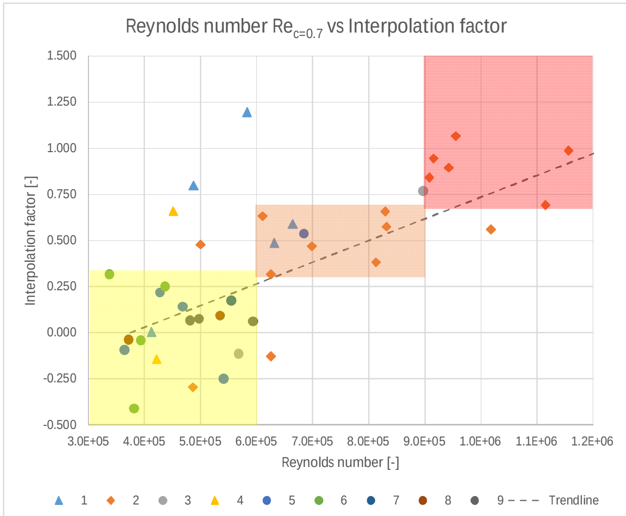

Figure 4: evaluation of Interpolation Factor as function of local Reynolds nomber of 0.7R chord length

Three different regions are drawn, based on the value of the InterpolationFactor; the yellow region shows the values below 33%, indicating results more dominated by laminar flow and the red region shows the values above 66%, indicating turbulent flow. The intermediate zone is shown in orange. The horizontal borders are based on the Reynolds numbers, where the maximum Reynolds number with laminar flow is set to 6.0×10^5. From Reynolds numbers of 9.0×10^5 the flow is assumed to be turbulent. With this simple approach a large number of points have been covered by the three zones. However, there are some clear outliers, with a relative low Reynolds number but high InterpolationFactor score. A closer analysis of these points learns that the institutes 1 and 4 seem to have a more turbulent flow than others, which might be related to the way of working in the specific institutes.

Based on this approach, the Reynolds number, as defined in equation (2), can be used to divide the population into three groups.

Figure 5 shows the comparison of the measured and calculated open water efficiency for the propeller designs with a Reynolds number below 6.0×10^5. The large majority of points is within 1% bandwidth, covering a large range of designs, with variations in mean pitch, blade area ratio among others. Four points can be identified which are clearly outside the 1% bandwidth. The fact that these points lie below the line, indicate a lower measured efficiency than calculated with the laminar flow regime. It is there not surprising that the four points, which are outside the yellow zone, are now found below the line.

Figure 6 shows a similar diagram, where the propeller designs with Reynolds number above 9.0×10^5 are selected. In this case the points are as well in the 1% bandwidth.

From the two diagrams it can be concluded that the applied method can give accurate open water results, when the impact of the local Reynolds number of the 0.7R chord length is considered in the selection of the flow regime.

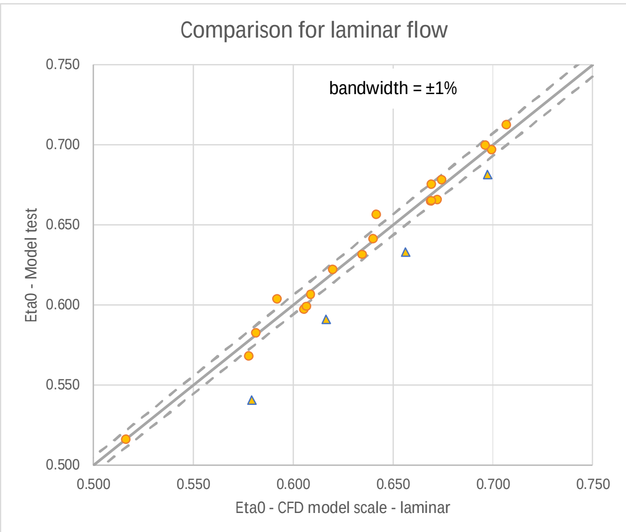

Figure 5: comparison of open water efficiency at design point between CFD simulations based on laminar flow regime and model scale experiments for propellers with $Re_{c-0.7}$ below 6.0×10^5

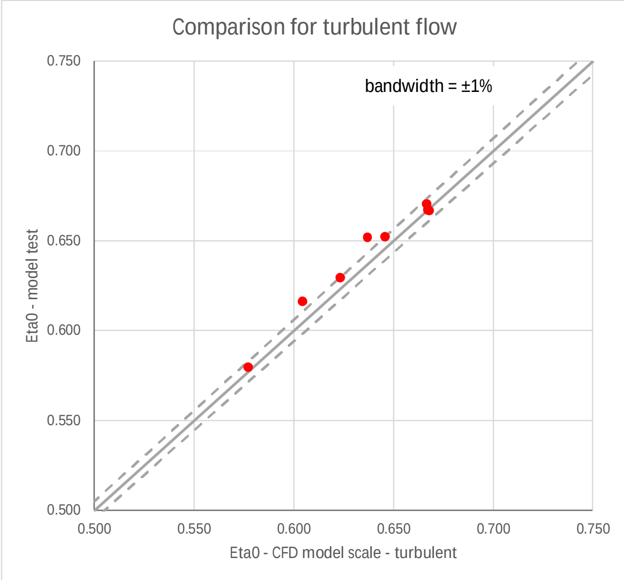

Figure 6: comparison of open water efficiency at design point between CFD simulations based on turbulent flow regime and model scale experiments for propellers with $Re_{c-0.7}$ above 9.0×10^5

For the actual full scale propellers, it can be assumed that the flow behavior of the great majority will be in the turbulent regime. The comparison of the open water efficiency between the model scale and full scale results is shown in Figure 7. A clear difference in scaling effect can be observed between the propellers with a laminar flow regime on model scale and the propellers with turbulent flow. The average improvement in open water efficiency for the propellers with laminar flow is around 2.3%, based on the current population. This Reynolds scaling impact is quite well in line with the scaling as applied by the various model basins, which is around 2.5% (Bulten, 2016). The propellers with turbulent behavior on model scale show an average increase of 6.9% at full scale. The difference in open water efficiency can be attributed to two different phenomena:

- Change in wall friction coefficient
- Change in flow patterns due to impact of centrifugal forces in boundary layer

For the change of the friction coefficient a correction methodology could possibly be developed, based on the contribution of the frictional forces, due to the flow similarities between turbulent flow at model scale and full scale. On the other hand, a clear change in flow behavior from laminar to turbulent flow has been observed in both model tests (Kuiper, 1981) and CFD model scale simulations (Bulten, 2015). Due to the lack of flow similarity, the results from laminar flow cannot be transformed into an equivalent model scale turbulent performance estimate.

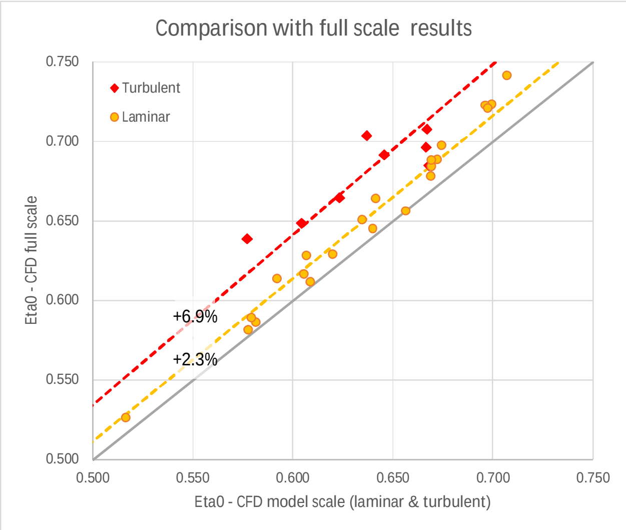

Figure 7: Reynolds scaling effect for propellers with laminar and turbulent flow regimes on model scale

4.2 DUCTED PROPELLERS

The CFD simulations for ducted propellers have been validated to a large extend with recent model scale measurements of the Wageningen D-series propellers (see Dang 2012). To secure the confidentiality of the results of this Joint Industry Project, no details about the validation process will be shown in this paper. However, the results as shown in Figure 8 for another propeller in Wärtsilä HPN nozzle, are representative for the larger population of ducted propellers analyzed with CFD. The model scale simulations are carried out based on a turbulent flow regime. Even in case the actual Reynolds number at 0.7Radius is below $9.0x10^5$, the velocity gradients at the entrance of the duct, seem to reduce the laminar flow behavior of the propeller.

The difference in open water efficiency in free sailing condition for the analyzed ducted propeller is about 10%. This fairly large Reynolds scaling impact is in line with the analysis as made for the bollard pull performance of ducted propellers (Bulten & Nijland, 2011).

The actual Reynolds scaling impact of a ducted propeller propulsion unit depends on the applied nozzle and propeller design. A comparison of two nozzle types is shown in Figure 9. For this comparison the propeller pitch has been adjusted to get a comparable open water efficiency at full scale. In this way the differences in performance on model scale are made clearly visible.

The actual Reynolds scaling effects as found for the two nozzle types are 10% and 14%. The nozzle with the lower model scale performance is equipped with a diffuser section, which seems to increase the scaling effects. At model scale, the risk for flow separation near the exit is larger, which reduces the effectiveness of the curved diffuser shape.

From model scale point of view, one can also argue, that modern nozzles with diffuser exit shapes will have even a larger positive scaling effect than conventional nozzle types.

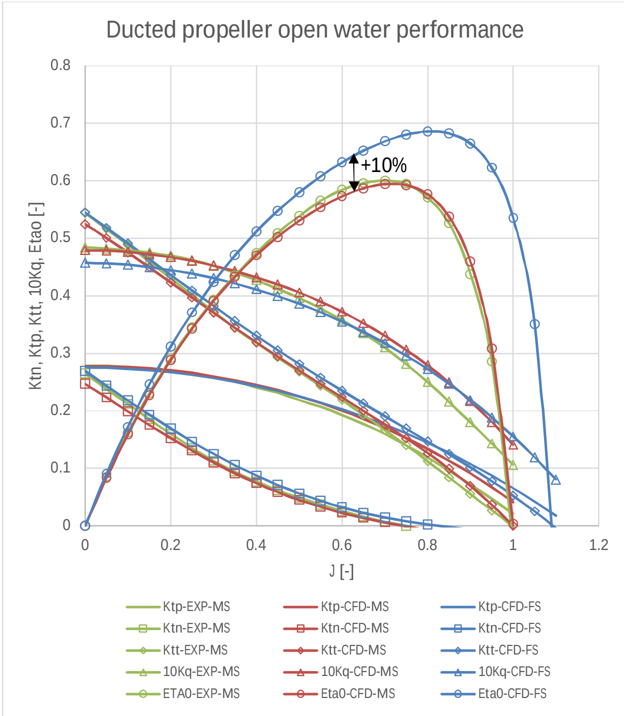

Figure 8: open water performance for ducted propeller based on model scale measurements, model scale CFD simulations and full scale CFD simulations

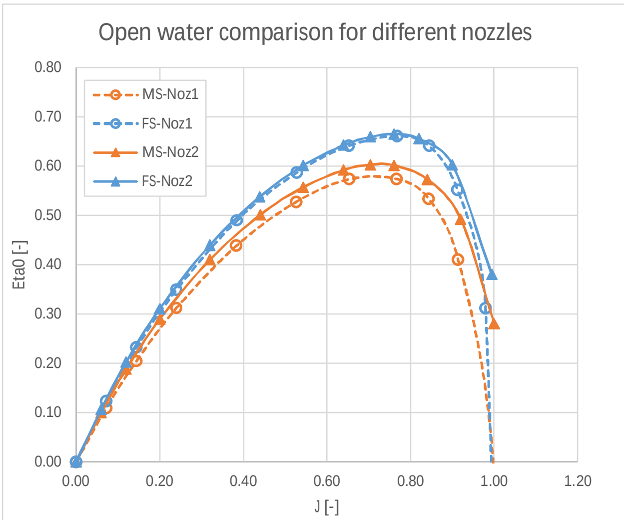

Figure 9: comparison of Reynolds scaling impact for 2 different nozzles

4.3 PUSHING THRUSTERS WITH DUCTED PROPELLERS

The open water performance of an azimuth thruster with fixed pitch propeller and nozzle has been presented in 2013 (Bulten & Suijkerbuijk, 2013). On model scale good agreement was found with model scale measurements, based on fully turbulent flow regime. The comparison of model scale and full scale open water performance is shown in Figure 10. In transit condition, an efficiency improvement of about 13-15% can be reached. This significant improvement can be partly attributed to the scaling of the ducted propeller performance, comparable to the ducted propellers. Another contribution comes from the reduced resistance of the thruster gearbox housing and the strut.

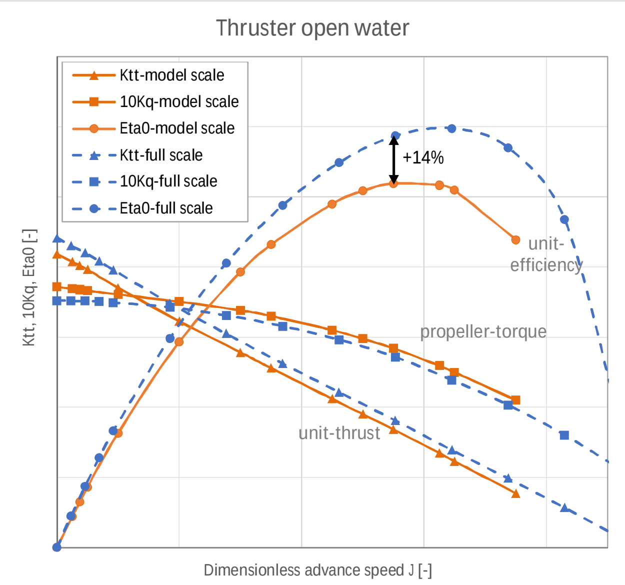

Figure 10: open water performance of azimuth thruster for model scale and full scale

4.4 CONCLUDING REMARKS ON OPEN WATER SCALING EFFECTS

In the previous sections, the Reynolds scaling effects of different propulsion configurations have been analyzed.

Propulsion type	Open water performance increase
Open propeller ($Re_{c-0.7}$ < $6.0x10^5$)	~2%
Open propeller ($Re_{c-0.7}$ > $9.0x10^5$)	~7%
Ducted propeller	10%-14%
Azimuth thruster (ducted propeller)	~14%

Table 1: Open water performance efficiency gains in free sailing mode for different propulsion configurations

In the table the summary of the observed scaling effects in free sailing (transit) mode are listed. The increase in performance increases in line with the complexity of the propulsion unit.

5 EVALUATION OF CAVITATION BEHAVIOUR

The differences in flow behavior are not limited to the open water performance only. In this section a comparison of the observed cavitation behavior at model scale and full scale will be presented. First the model scale CFD simulations will be compared to the observed cavitation pattern in the model tests. Thereafter the comparison between the CFD results at model and full scale will be made.

5.1 CAVITATION AT MODEL SCALE

CFD simulations with a cavitation model activated have been presented in detail at the previous SMP conference (Bijlard & Bulten, 2015), where detailed results of the validation process have been discussed. Model scale experiments have been carried out for a 25 cm propeller with 20 Hz rotational speed. This condition represents a condition $Re_{c-0.7} = 1.0x10^6$. Agreement between observations from the cavitation tunnel experiments and the CFD simulations were good. Moreover the differences between the model scale cavitation pattern and the full scale cavitation were limited. Based on the calculated Reynolds number and the level of free-stream turbulence in the tunnel, it is most probable that the observed flow behavior is based on the turbulent regime. It should be noted that the tested thruster unit is of relatively large size (scale factor l = 15.6), and the propeller rotational speed has been quite large as well.

In another set of model scale tests, where a thruster has been tested in self-propulsion configuration in a vacuum tank, a lower Reynolds number had to be accepted. These tests gave significantly more cavitation on the propeller blades than expected when the blade passed the strut of the thruster.

The observed cavitation (top) and the calculated cavitation shape (bottom) are shown in Figure 11 (top) for the same blade position. The amount of cavitation at the lower radii is not in line with the expectations based on the design software.

More detailed analysis of the flow field, as calculated with CFD, revealed a large wake behind the strut due to laminar flow effects. The low velocity into the propeller results in increased blade loading and consequently more cavitation. This phenomenon is shown even more pronounced in the upper picture in Figure 12, where the worst case blade position has been selected.

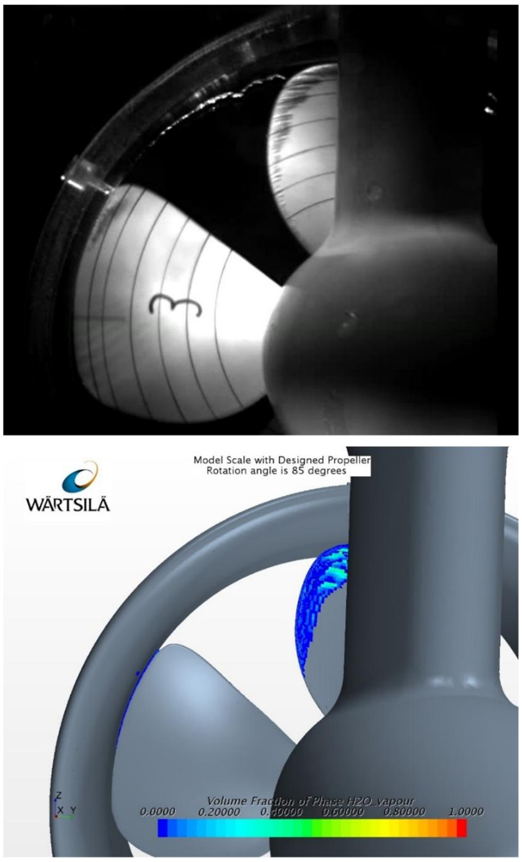

Figure 11: observed cavitation (top) and calculated cavitation on model scale

5.2 CAVITATION AT FULL SCALE

The cavity shape, as shown in the top of picture of Figure 12, would in general not been accepted and a redesign of the propeller blade would have been made. However, for this design a full scale CFD simulation has been made as well. The result for the full scale (realistic) operating condition is shown in the bottom picture of Figure 12.

The amount of cavitation at full scale is very limited and certainly acceptable. Based on these findings it has been concluded that the existing design is well fit for purpose. Moreover, any modification of the leading edges of the propeller blades, could improve the cavitation on suction side, but at the same time it would cost margin against cavitation at the pressure side. So, in case a redesign was considered, the risk for pressure side cavitation erosion could have been increased significantly.

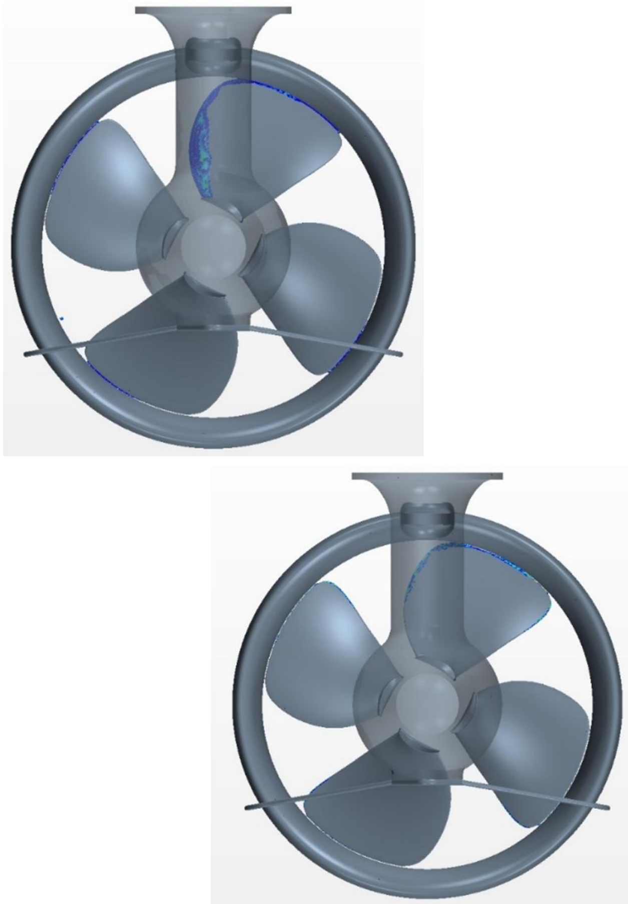

Figure 12: calculated cavitation pattern for model scale (top) and full scale

6 PERFORMANCE IN BEHIND SHIP CONDITION

So far the impact of Reynolds scaling effects have been limited to the open water performance predictions and the cavitation behavior due to the flow disturbance of the thruster strut. The final performance estimate of a vessel is based on the performance of the propulsion unit in behind ship condition however. Detailed impact analysis of the Reynolds scaling impact is currently still work in progress, but some general ideas will be shared in the following sections already.

6.1 TRANSITION FROM LAMINAR TO TURBULENT REGIME

Many experts in the marine industry expect that the Reynolds number is the major driving factor towards either laminar or turbulent flow regime of the flow. Based on this assumption and the knowledge that the propeller RPM is in general larger in the open water test compared to the self-propulsion condition, a turbulent flow regime is expected in the open water tests and possibly as well in the self-propulsion test. However, if the free stream turbulence is regarded to be the driving force, a turbulent flow regime in the self-propulsion tests is likely, due to all flow disturbances in the wakefield. This approach supports the idea of a laminar flow regime in the open water tests, since those can be operated with low levels of free stream turbulence.

6.2 VALIDITY OF ITTC'78 EXTRAPOLATION METHOD

The method for propeller open water scaling as proposed in the ITTC'78 procedure need to be revisited, based on the current level of knowledge and understanding of the occurring flow phenomena. The performance scaling for propellers is based on a correction on both Kt and Kq of the propeller, with a relation to the friction factor. Since there is a clear lack of flow similarity between model scale laminar flow and full scale turbulent flow, the simplicity of the ITTC'78 method can be challenged.

Besides the performance scaling of the propeller, a number of other flow phenomena are being addressed in the extrapolation method. Due to the large variety of factors which are considered in the extrapolation methods, it is very well possible that different errors cancel each other. This concept is shown in Figure 13, where the probability out of an accurate full scale estimate is sketched. At this moment, it is still expected that the current full scale predictions based on extrapolation methods give on average an accurate prediction. There is a certain spread around the optimum prediction, which is a result of the spread introduced by the extrapolation method (ways how errors cancel each other) and the natural spread in the model scale measurements.

The full scale CFD simulations are characterized by a much smaller spread, assuming that the internal quality assurances processes are in place. Still there might be an off-set to the actual accurate prediction, due to lack of high quality full scale validation material. Nevertheless, the potential of the full scale CFD simulations is more positive, due to the low spread. This low spread is an important feature in the current search for energy efficiency and the design of all kind of energy saving devices. For performance improvements in the range of 1-2%, an analysis method with significantly lower spread is required to be able to make proper distinctions between differences in performance due to design or due to measurement spread.

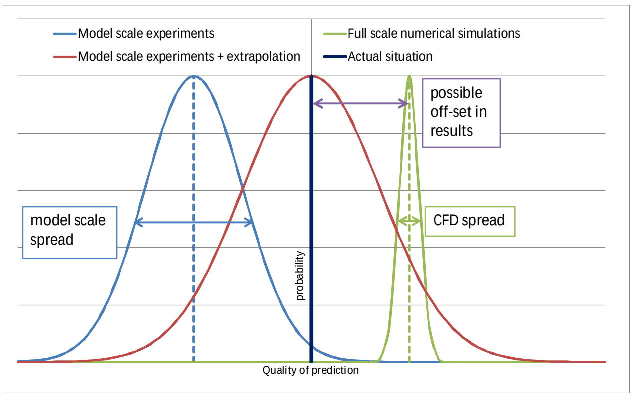

Figure 13: sketch of probability of accurate full scale performance predictions

7 CONCLUSIONS

The developments of the viscous numerical flow simulations during the last decade have resulted in a clear improvement of the accuracy of the performance predictions of various available types of propulsion units, like propellers, ducted propellers and azimuth thrusters. With a well thought meshing strategy for model and full scale and the accurate results from validation studies, sufficient confidence in the full scale results can be obtained for more detailed analyses. Comparison of the performance on model scale and full scale has learned that the commonly applied ITTC'78 extrapolation method seems to be valid for the conventional open propellers, but there is room for debate when ducted propellers and thruster units are considered. Although the method seems to be capable of predicting the expected open water performance gain for open propellers, possible differences between laminar and turbulent flow regimes are not explicitly captured in the extrapolation methodology.

The presence of laminar flow effects in behind of the strut of an azimuth thruster can have a dramatic effect on the cavitation behavior of the propeller. With the tight design margins of modern propulsion units, it is often not possible anymore to make designs which can perform well in both model scale experiments and actual full scale operation. It should be obvious that the design focus should be on the full scale operation.

Given the continuous development of the numerical flow simulations, it seems to be a matter of time until the direct full scale simulations will become the common approach. The main argument to stick to the conventional methodology (as shown in Figure 1) is the limited availability of full scale validation material. It is therefore of utmost importance to gather all kind of full scale validation and operational data from vessels sailing all over the world. With trends from the Big Data, it should be possible to get more insights in the existing off-set of the full scale numerical simulations (as shown in Figure 13).

It can therefore be concluded that the maritime industry should embrace the full scale numerical flow simulations and put effort in continuous improvement of the actual full scale (prediction of the) performance.

ACKNOWLEDGMENTS

Support for this research was provided by the Wartsila Propulsion CFD department and by mr. Navneet Chadha and mr. Vignesh Murali from Eindhoven Technical University.

REFERENCES

Bijlard, M., Bulten, N. (2015),' RANS simulations of cavitating azimuthing thrusters', Proceedings 4th International Symposium on Marine Propulsors, Austin, TX

Bulten, N., Nijland, M.(2011),' On the development of a full-scale numerical towing tank', Proceedings 2nd International Symposium on Marine Propulsors, Hamburg, Germany

Bulten, N.W.H., and Suijkerbuijk, R. (2013), "Full scale thruster performance and load determination based on numerical simulations", Proc. 3rd International Symposium on Marine Propulsors, Launceston, Australia.

Bulten, N. (2016), 'With numerical simulations to more efficient ship designs', Proceedings RINA Energy Efficient Ships conference, London, UK

Bulten, N., Stoltenkamp, P. (2017),'Improved sustainable speed due to thrusters with ducted propellers, Proceedings of the ASME 2017 36th International Conference on Ocean, Offshore and Arctic Engineering OMAE2017, Trondheim, Norway

Dang, J., Brouwer, J. Bosman, R., Pouw, C. (2012),' Quasi-steady two-quadrant open water tests for the Wageningen Propeller C- and D-Series', Proceedings of 29th Symposium on Naval Hydrodynamics (ONR), Gothenburg, Sweden

Guiard, T., Leonard, S., Mewis, F. (2013),'The Becker Mewis Duct – challenges in full-scale design and new developments for fast ships', Proc. 3rd International Symposium on Marine Propulsors, Launceston, Australia.

Korkut, E., Atlar, M. (2002), 'On the importance of the effect of turbulence in cavitation inception tests of marine propellers', Proc. Royal society **458**, pp 29-48

Kuiper, G. (1981),'Cavitation inception on ship propeller models', PhD thesis, University of Delft, The Netherlands

Lafeber, F.H., Brouwer, J. Dang, J. (2013), 'A quasi-steady method for efficiently conducting open water model tests', Proceedings 3rd International Conference on advanced model measurement technology, Gdanks, Poland

Ponkratov, D., Zegos, C., (2015),'Validation of ship scale CFD self-propulsion simulation by the direct comparison with sea trials results', Proceedings 4th International Symposium on Marine Propulsors, Austin, TX

Sánchez-Caja, A., González-Adalid, J., Pérez-Sobrino, M., Sipilä, T. (2014),'Scale effects on tip loaded propeller performance using a RANSE solver', Ocean Engineering 88 (2014) pp 607-617

DISCUSSION

Question from Sverre Steen / Julie Yang

How do you account for roughness in your full scale CFD? If you don't account for roughness, how do you think that impacts the model-full scale ratios you show?
Have you considered the influence of manufacturing tolerances and measurement accuracy?

Author's closure

Roughness is indeed an import issue which needs to be implemented still. Application of roughness in RANS simulations is not straightforward however. Proper validation of the methodology is thus required.
Roughness on ship hull surface can be easily in the range of few hundreds of microns. Propellers are polished to surface roughness of a few micron, depending on finishing class. Impact on resistance might therefore be of different magnitude than the (initial) performance of the propeller.
Related to this is the impact of manufacturing tolerances. The allowed tolerances based on ISO 484 procedure, will result in a spread of performance, which can be few tenth of percent's in overall performance.
The measurement accuracy could be related to the actual manufactured propeller, which is a challenging task to confirm that the final produced propeller is matching the designed geometry. With modern laser measurement techniques the manufacturing quality can be verified sufficiently.
Measurements of the performance on either controlled conditions in model scale basins or at full scale are both suffering from less or more spread. It is however the intention to use the information of the CFD simulations to steer the design process. In that phase it is more of importance to be able to distinguish between better and worse performing designs in relative sense than on the actual absolute value.

Continuous improvement of the numerical methods will bring the results closer to the absolute thrust over time.

Question from Stephen Helma

You investigated scale effects on propellers in nozzles. Which kind of nozzle were these: accelerating or decelerating? Are there any differences in the scaling magnitude of these two types of nozzle?

Author's closure

The CFD simulations so far have been applied to the accelerating nozzles, where the velocity is increasing towards the propeller. On the other hand, a lot of work has been done at Wärtsilä on waterjets, which can be regarded to a certain extend as decelerating nozzles. In all devices the observed Reynolds scaling effects can be directly linked to the scaling of the pumping efficiency, which has been identified in the first part of last century.

Question from Chen-Jun Yang

If there is a necessity to perform CFD-based scaling for ducted propellers, do you think it is necessary to scale the tip clearance? How to validate the results/method in that case?

Author's closure

The tip clearance is indeed a very important parameter for the overall performance of the ducted propeller. Research at Wärtsilä has learned that a smaller tip clearance (<1%) improves the overall performance. This phenomenon is utilized in the steerable thrusters with fixed pitch propellers, where a tip clearance of 0.5% is applied.
Based on dimensionless analyses and similarity rules, the propeller tip clearance should be scaled geometrically. This would mean a tip clearance of 0.5 mm between blade and nozzle for a 200 mm propeller in such case.