Return to Session Directory

DYNAMIC POSITIONING CONFERENCE

October 17-18, 2006

Thrusters

Numerical Analysis of Flow Around a Thruster

Norbert W. H. Bulten

Wärtsilä Propulsion Netherlands (WPNL)

The Netherlands

Introduction

Recent developments in thruster design show an increasing interest in the application of thrusters at relatively high speeds and in higher required bollard pull. For free sailing condition this means a high free sailing open water efficiency of the unit. Performance of the bollard pull can be expressed in a so-called merit-coefficient (mc). This coefficient gives the non-dimensional ratio between thrust and torque of an installation. A high Merit coefficient represents a high bollard pull.

Steerable thrusters are available in a wide variety of configurations. The main components are in all configurations a fixed or controllable pitch propeller, a gearhouse and a strut or steering pipe. Many thrusters are equipped with a nozzle as well. The location of the propeller with respect to the strut results in a pushing or pulling type thruster.

The main differences with a single open or ducted propeller are the presence of the gearhouse and the strut. The effect of these two parts on the overall performance is twofold. Firstly, these parts can have a significant drag, which reduces the total unit thrust and secondly the inflow field to the propeller is affected by the strut especially in case of a pushing type unit. This can result in a change of the operating point of the propeller.

Research and development of propellers has been based on model scale measurements for decades. An interesting result of the experiments has been the development of a set of nozzles by MARIN, of which type 19A is the most commonly used (see for example Van Manen, 1957). Model scale experiments suffer from Reynolds scaling effects, which are compensated for by most model basins by means of empirical corrections in the extrapolation procedure. These scaling methods are based on normal ducted propellers in general. Scaling methods for thruster units are not straight forward. This can be attributed to the large variety of thruster configurations to a certain extent. Another aspect is the lack of understanding of the nature of the occurring flow phenomena and forces on the thruster unit. It is impossible to derive a good scaling method for a thruster unit without a good knowledge of the occurring flow phenomena.

The need for Reynolds scaling methods is eliminated, when the research is based on viscous numerical methods. These methods can be used to calculate the flow around a thruster unit both on model scale and at full scale. During the last decade the use of these methods, denoted as RANS methods, has increased enormously in industry. At Wärtsilä Propulsion Netherlands the numerical approach is used for many years to design the inlet geometry of waterjet propulsion systems (Verbeek&Bulten, 2001) and to determine open water performance characteristics of open and ducted propellers for example. One of the interesting conclusions of the calculations is the determination of the sensitivity of scaling on the bollard pull performance of the Lips-HR-nozzle. Calculations on both model scale and full scale have proven that there is a significant scaling effect on the bollard pull performance of the nozzle. This phenomenon will be discussed in more detail in this paper.

The numerical method will be used to investigate the effect of the gearhouse and the strut on the open water performance of the thruster unit. Calculations are made on both model scale and full scale. The results of the model scale calculations are used for comparison with experimental data. In this way the accuracy of the numerical method is verified.

The results of the full scale thruster unit can be used to make a detailed investigation of the occurring flow phenomena. The forces, acting on the various parts of the thruster unit, like the propeller, nozzle and gearhouse, can be evaluated separately. This gives a clear picture of the thrust and drag forces acting on the separate parts. It will be shown that the actual drag of the thruster house can be divided in a component related to advance speed and a component related to propeller thrust. The latter is important at bollard pull conditions.

Background of numerical method

The set of equations that describe all flow phenomena in general are the Navier-Stokes equations. These equations can not be solved numerically for practical applications. Simplifications are made to obtain a set of equations that can be solved with normal computers nowadays. The applied type of simplification is related to the required computational time in general. Three typical numerical methods can be distinguished:

1. potential flow methods
2. Euler flow methods
3. Reynolds averaged Navier-Stokes methods

The first two methods do not take the effects of viscosity into account and these are therefore not suitable to investigate Reynolds scaling effects. Such a study can be made with the third method, where the effects of viscosity are included. The simplification of the original Navier-Stokes equation is found in the treatment of the turbulence of the flow. The chaotic behavior of the flow is modeled by a so-called turbulence model.

The numerical model is used to solve the velocity field and the pressure distribution in a 3-dimensional volume around the thruster unit. The complete domain is divided into a large number of cells, denoted as the mesh. At the surrounding of the numerical domain boundary conditions are prescribed. Typical examples of these boundary conditions are inflow boundary conditions and constant pressure boundary conditions.

Figure 1 shows an example of the surface mesh of a complete thruster unit and the total numerical domain, which includes the thruster. The calculations provide the three velocity components and the pressure in each cell. This enables the calculation of the pressure and shear forces acting on the surface of the thruster unit.

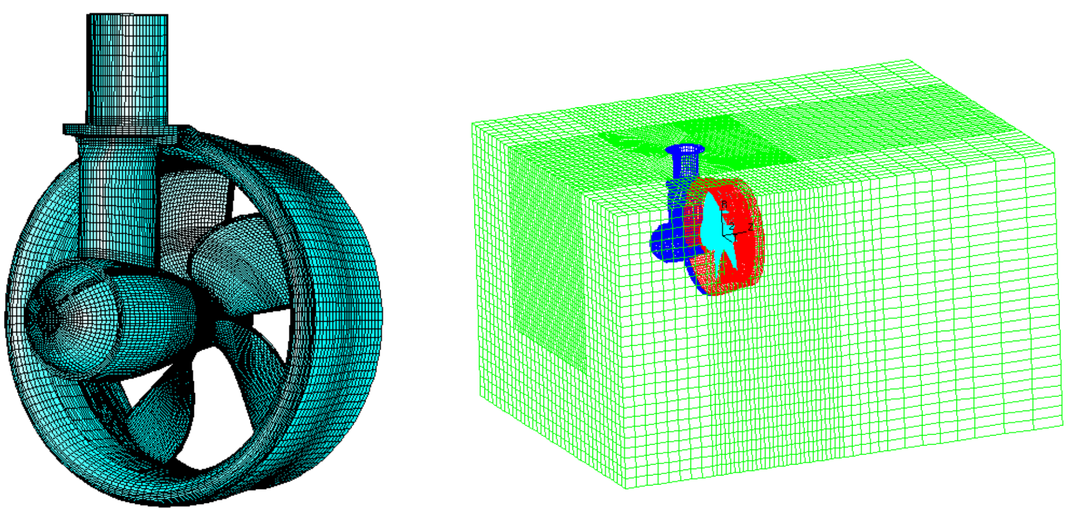

Figure 1: Example of surface mesh of complete thruster unit (left) and mesh of numerical domain.

Validation of numerical method

The complexity of the numerical model requires a detailed validation process before the results can be used for further analyses. Validation of the method can be done with aid of experimental data from model scale measurements. In this paper the results of the calculation of the open water performance of a KA5-75 propeller in a 19A nozzle are compared with the available experimental data, as shown in figure 2.

Agreement between the experimental data and the calculated thrust is very good over the complete range of J values. The prediction of torque shows larger deviation between measured and calculated values, but these differences are still acceptable. This is a typical result for RANS methods, which is found for a large variety of open and ducted propellers. The background of this deviation is discussed in more detail in the paper of Bulten&Oprea (2005).

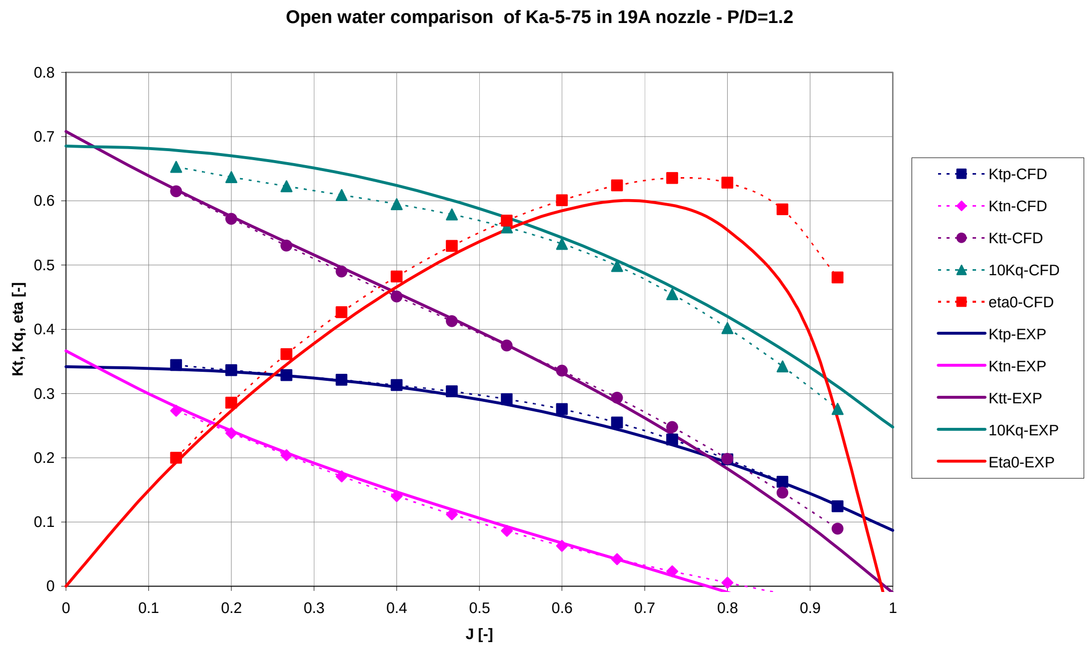

Figure 2: open water performance comparison of Ka5-75 in 19A nozzle

Reynolds-scaling effects of nozzle performance

The numerical method takes viscous effects into account. This enables the comparison of results of model scale and full scale calculations to determine Reynolds scaling effects. The calculations are made for both the 19A and the HR-nozzle with the Ka5-75 propeller. The cross-sectional profiles of both nozzles are shown in figure 3.

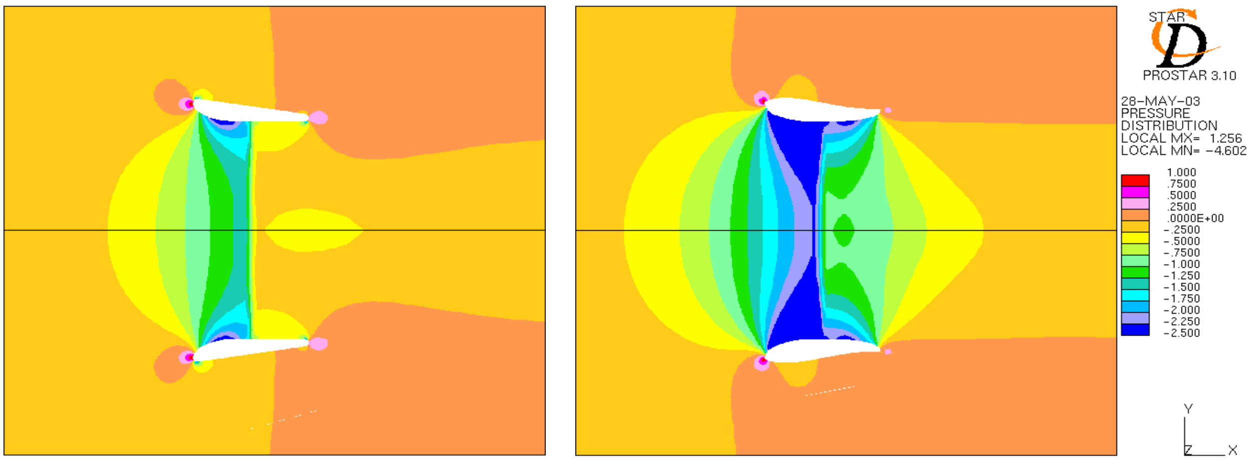

Figure 3: Comparison of 19A (left) and Lips-HR nozzle cross-sectional profile. Contours indicate pressure distribution in free-sailing condition

Figure 4 shows the open water performance for the Ka-propeller in a 19A nozzle. It can be seen that the difference in efficiency is rather limited for this configuration, though both thrust and torque show an increase. The calculations for the Ka-propeller in a HR-nozzle show a more significant difference between model scale and full scale. This is attributed to the improved thrust of the HR-nozzle at free sailing conditions. The maximum open water efficiency improves at full scale with about 7% compared to model scale.

Performance at bollard pull condition can not directly be evaluated from the open water diagrams. For this purpose the so-called merit-coefficient (mc) is used. This coefficient is defined as:

$$mc = \frac{\left(Ktt/\pi \right)^{3/2}}{Kq}$$

With this equation a performance indicator for J=0 condition is created. The result for the merit coefficient of the Ka-propeller in a HR-nozzle is shown in figure 6 for both model scale and full scale. At bollard pull condition the full scale merit coefficient is about 10% higher than at model scale, which results in an improved bollard pull of about 7% for equal power.

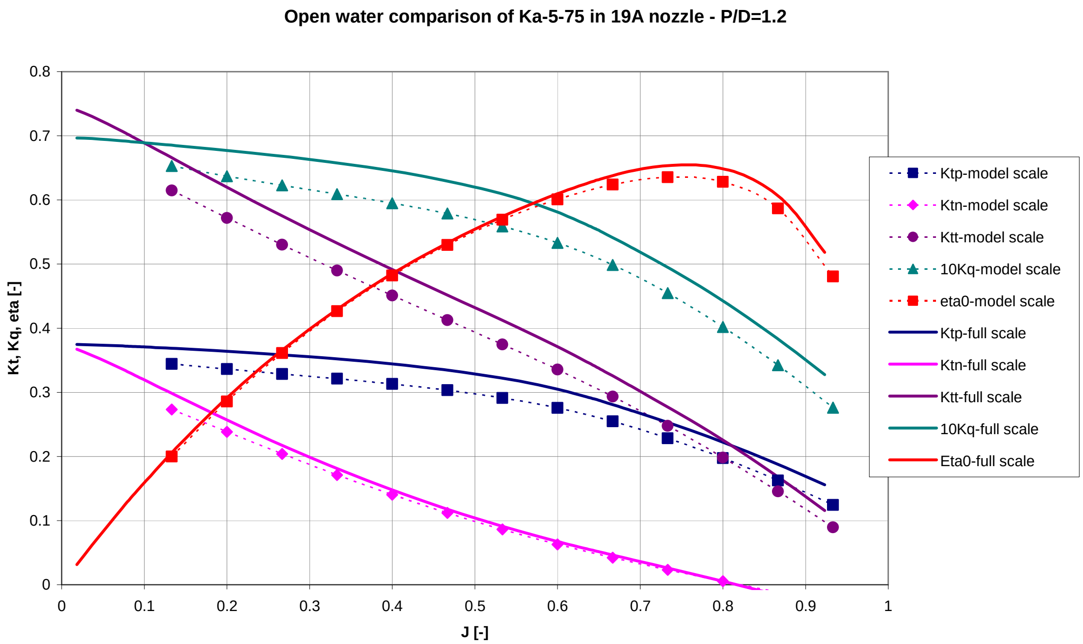

Figure 4: Comparison of open water performance of Ka5-75 propeller in 19A-nozzle on model scale and full scale.

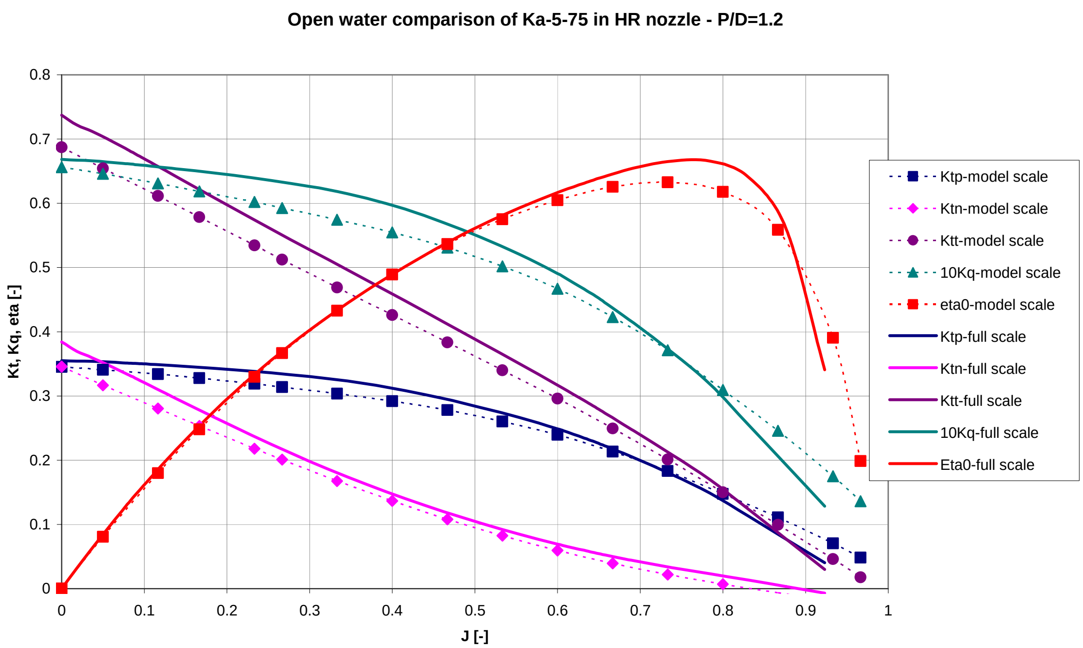

Figure 5: Comparison of open water performance of Ka5-75 propeller in HR-nozzle on model scale and full scale.

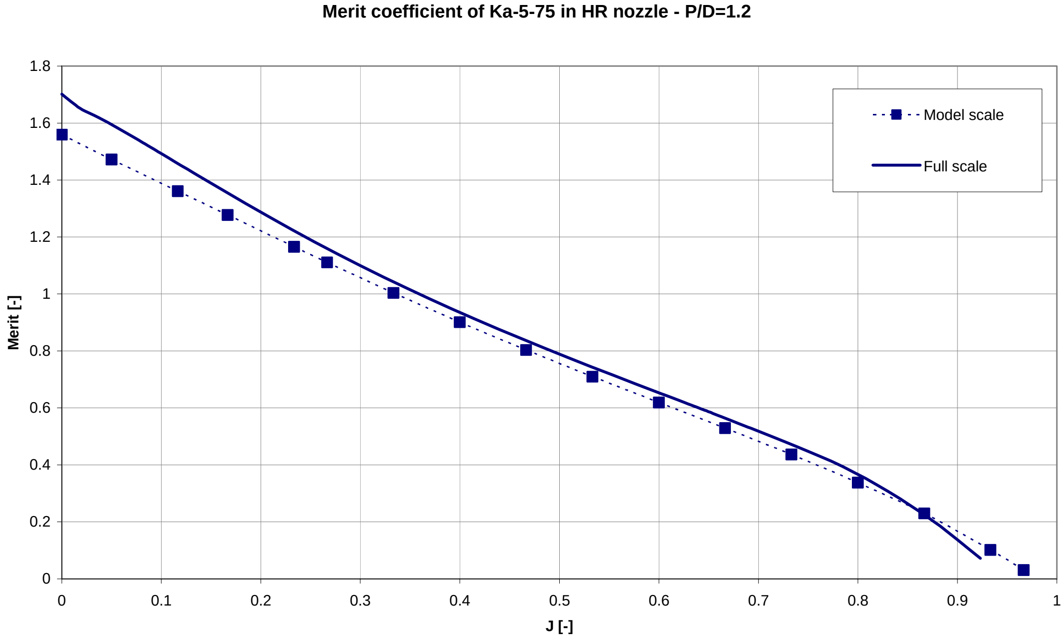

Figure 6: Merit coefficient of Ka5-75 propeller in HR-nozzle on model scale and full scale.

From the numerical results it is concluded that the Reynolds scaling effect of a full scale propeller with HR-nozzle is significant. Predictions of the open water performance should be based on numerical analyses preferably to avoid the problem of scaling.

Influence of thruster-housing on performance

In the previous section the effect of Reynolds scaling effects on a nozzle is discussed. Those results are based on calculations with a propeller-nozzle configuration with a hub used in a standard open water test at model scale, further referred to as "test-hub". The presence of the thruster-housing (gearhouse and strut) will have a different influence on the overall open water performance of the thruster unit. Figure 7 shows the free sailing pressure distribution for a test-hub configuration and a complete thruster unit to show the differences between the two geometries.

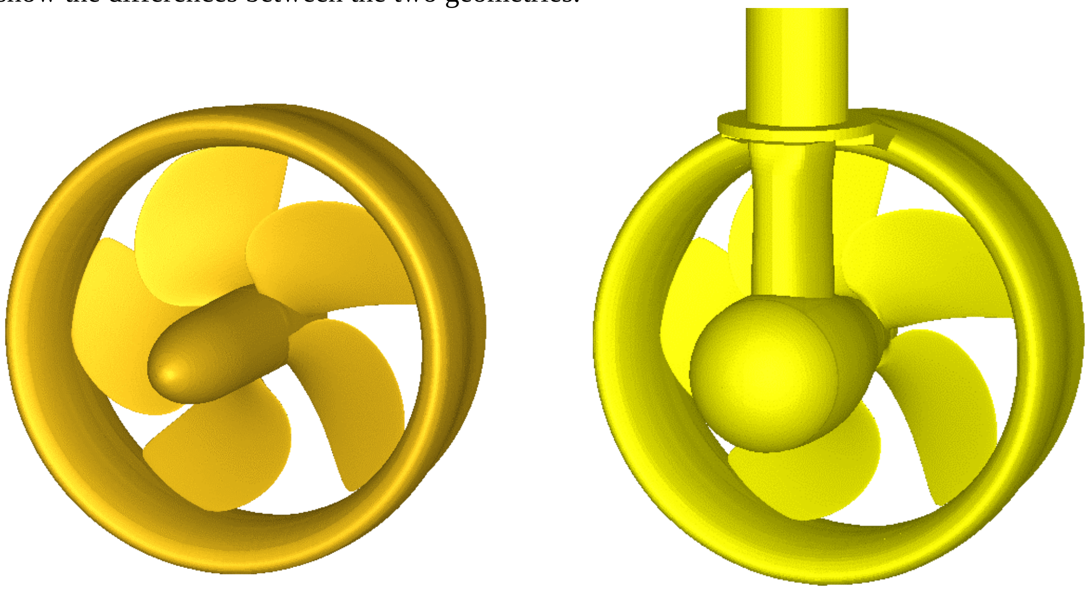

Figure 7: Comparison of test-hub configuration (left) and thruster unit

The open water performance of both configurations is compared in figure 8 for full scale Reynolds number. This figure shows the unit thrust, propeller torque and the overall unit efficiency. It can be seen that the unit thrust is the same for both configurations at bollard pull condition (J=0). For increasing speed the deviation between the two increases. The total unit thrust of the thruster is lower compared to the test-hub configuration.

The comparison of the torque shows a difference in behavior between bollard pull condition and free sailing condition. At bollard pull, the torque of the thruster unit is higher than the torque of the test-hub propeller. For free sailing conditions the highest torque is found for the test-hub.

The maximum efficiency of the thruster-unit is lower than the test-hub unit, which is in accordance with the expectations. However, the J-value of the maximum efficiency is reduced as well for the thruster unit. This phenomenon should be kept in mind when a thruster propeller is designed or tested on model scale. Selection of a propeller design, based on the open water test-hub best efficiency, will result in a thruster unit with an operating point beyond the best efficiency.

The Reynolds scaling effect on the HR-nozzle has been discussed in the previous section for the test-hub configuration. It is to be expected that the efficiency improvement is found as well for a thruster unit. The open water performance comparison for a model scale and full scale thruster unit is shown in figure 9.

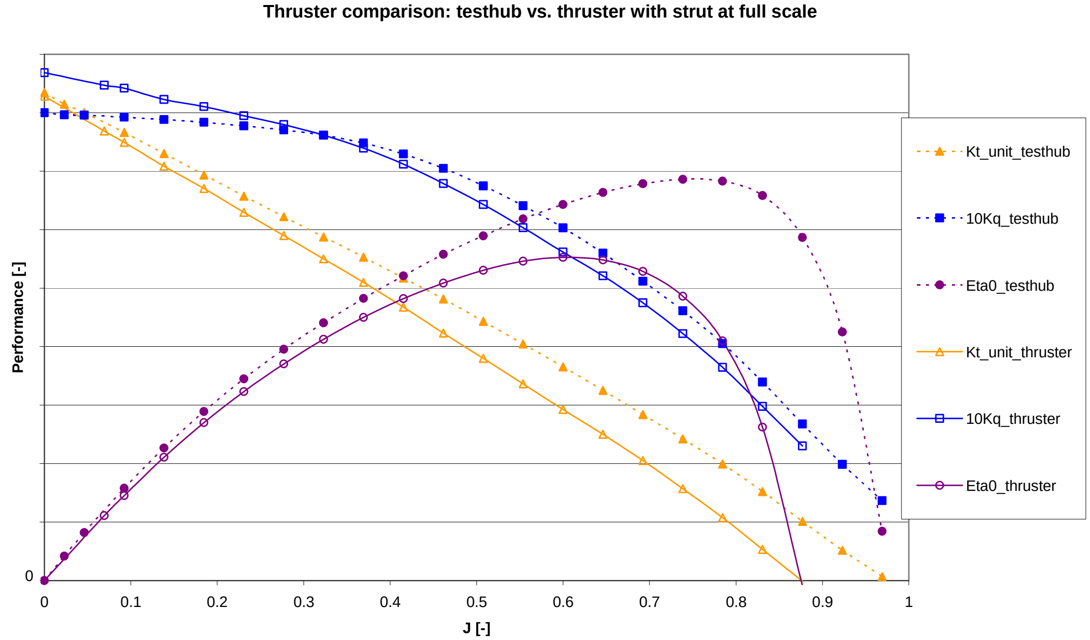

Figure 8: Comparison of open water performance of test-hub configuration (dotted lines) and thruster unit (solid lines) for full scale

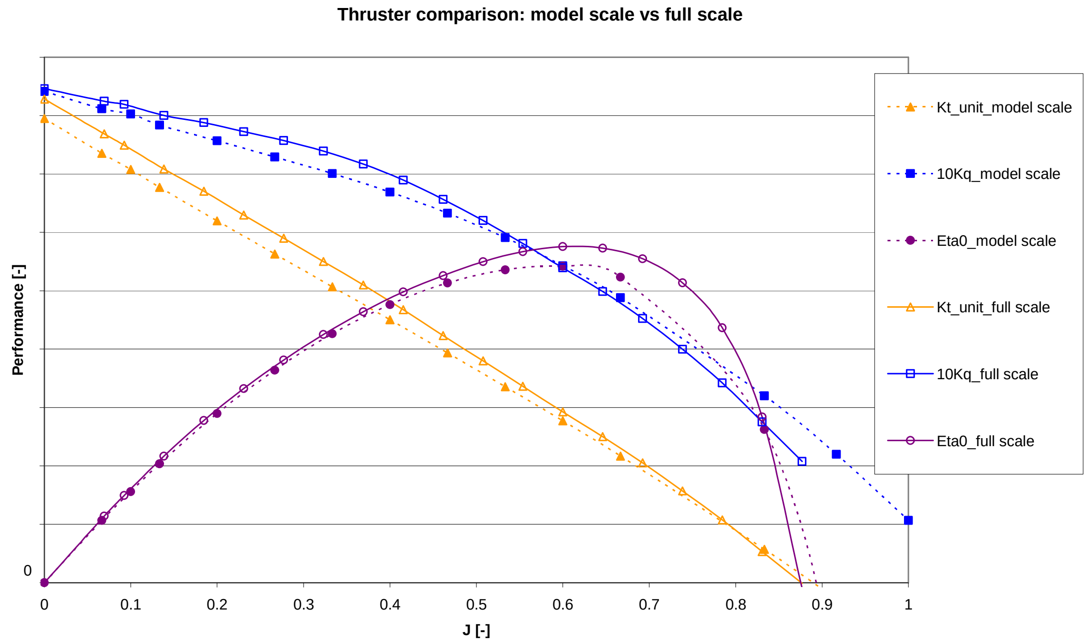

Figure 9: Comparison of open water performance of model scale thruster unit (dotted lines) and full scale thruster unit (solid lines)

For both bollard pull condition and at free sailing condition the torque is more or less identical for both scales, whereas the unit thrust shows a clear improvement for the full scale unit. The deviations between model and full scale for thrust and torque are not constant over the complete range of operating conditions however. Consequently, it is not straight forward to derive a simple Reynolds scaling method.

Drag of thruster strut

Analysis of the deviation between the total unit thrust of the thruster and the test-hub configuration learns that the loss in unit thrust increases with advance speed (see figure 8).

With the numerical results it has become possible to investigate the contributions of the individual parts of the thruster unit to the total thrust in detail. The overall thrust can be split in the propeller thrust, nozzle thrust, gearhouse drag and strut drag. The comparison of the propeller and nozzle thrust for the two configurations is shown in figure 10. The drag components of the gearhouse and the strut are combined and presented with an opposite sign, compared to the thrust components.

Results of the analysis of the various drag components of the thruster unit are presented in figure 11. This diagram shows the individual drag of the gear-house and the strut and the combined drag. A fairly good approximation of the drag value can be obtained with a curve-fit based on:

$$Kdt = C_1 \cdot Kt_{unit} + C_2 \cdot J^2$$

In this equation C_1 and C_2 are two constants. The constant C_2 can be related to a drag coefficient, for known dimensions of the thruster house.

At bollard pull conditions the main contribution to the drag comes from the thrust related part. On the other hand, at free sailing conditions the drag coefficient and the advance speed are the most important factors. Agreement between the CFD results and the curve-fit is good for both the bollard pull condition as well as for the free sailing condition.

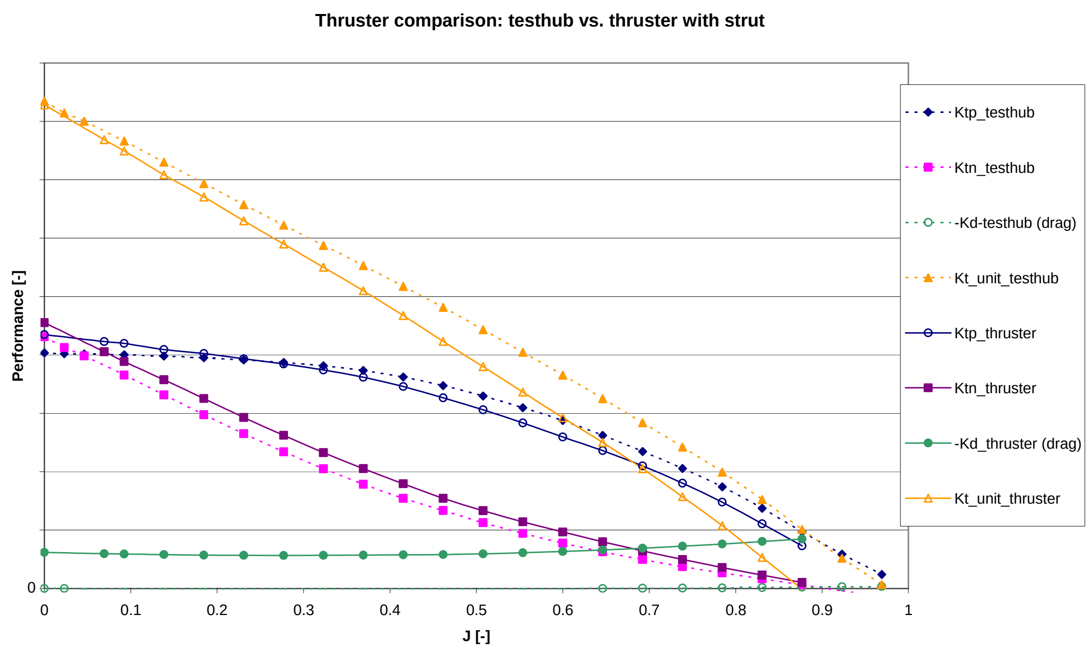

Figure 10: Comparison of propeller, nozzle and unit thrust of test-hub configuration (dotted lines) and thruster unit (solid lines)

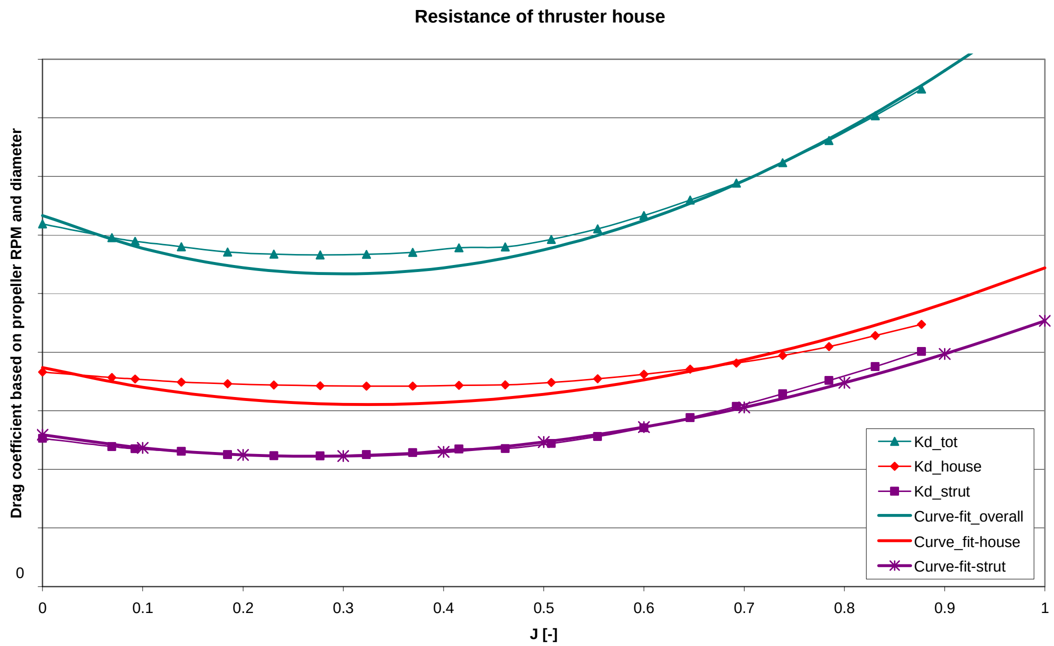

Figure 11: Analysis of thruster house and strut drag components

Conclusions

- The performance of the Lips-HR-nozzle is significantly better on full scale than on model scale. This is due to a large Reynolds scaling effect on the flow field around the nozzle. The performance of a ducted propeller can be determined accurately with a numerical viscous method.
- The presence of the thruster house and strut has an effect on the performance of the propeller. The best efficiency point of a thruster unit is found at a lower J-value than for the single propeller-nozzle configuration.
- The drag of a thruster house can not be expressed in a drag coefficient only. A much better prediction is found when the drag is related to both the propeller thrust and a drag coefficient. At bollard pull conditions the drag of the house is due to the propeller thrust contribution, whereas the drag coefficient becomes more important at free sailing conditions.
- Full scale numerical analyses seem to be most suitable for prediction of thruster unit performance. Extrapolation methods for model scale measurements can not be derived easily due to the complex nature of the thruster house drag.

References

Bulten, N.W.H. & Oprea, I.A., “Consideration on deviations in torque prediction for propellers and waterjets with RANS codes”, Proceedings RINA Marine CFD conference, Southampton, 2005

Van Manen, J.D. et al, “Recent research on propellers in nozzles”, Int. Shipb. Progress, Vol. 4, No 36, 1957

Verbeek, R. & Bulten, N.W.H., “Interpretation of model scale test results with aid on CFD calculations”, Proceedings RINA Waterjet propulsion III conference, Gothenburg, 2001

Appendix

Performance of the propeller and nozzle are expressed in the standard non-
Performance of the propeller and nozzle are expressed in the standard non-dimensional representation for advance ratio, thrust, torque and efficiency:

$$J = \frac{\mathrm{Vad}}{nD}$$

$$Ktp = \frac{\mathrm{Propeller\_thrust}}{\rho n^2 D^4} \qquad Ktn = \frac{\mathrm{Nozzle\_thrust}}{\rho n^2 D^4}$$

$$Ktt = \frac{\mathrm{Total\_thrust}}{\rho n^2 D^4} \qquad Kq = \frac{\mathrm{Propeller\_torque}}{\rho n^2 D^5}$$

$$\eta = \frac{\mathrm{J \cdot Ktt}}{2\pi Kq}$$

Drag coefficients are made non-dimensional in a similar way:

$$Kds = \frac{\mathrm{Strut\_drag}}{\rho n^2 D^4} \qquad Kdh = \frac{\mathrm{House\_drag}}{\rho n^2 D^4}$$