First International Symposium on Marine Propulsors
smp'09, Trondheim, Norway, June 2009

Scale effects on propellers for large container vessels

Sven-Brian Müller[1], Moustafa Abdel-Maksoud[2], Gerd Hilbert[3]

[1]Institute of Ship Technology and Transport Systems (IST), University of Duisburg-Essen (UDE)
[2]Institute of Fluid Dynamics and Ship Theory (FDS), Hamburg University of Technology (TUHH)
[3]Mecklenburger Metallguss GmbH (MMG)

ABSTRACT

The prediction of the performance of full-scale propellers is still depending on experimental investigation. The extrapolation of the measured thrust and torque data to full scale is affected by many factors, which influence the flow behavior on propellers. The importance of the problem increases with enlarging the used scale factor between full-scale and model.

Mecklenburger Metallguss GmbH (MMG) as one of the world-wide leading manufacturers of large propellers cooperates with the Institute of Ship Technology and Transport Systems (IST), University of Duisburg-Essen and the Institute of Fluid Dynamics and Ship Theory (FDS), Hamburg University of Technology, to investigate the scale effects on propellers. The research project is supported by the German Federal Ministry for Economics and Technology (BMWi).

Within this project the extrapolation problem of the thrust and torque coefficients from model to full-scale was analyzed by using the numerical calculation method for viscous flow ANSYS-CFX. To validate the used combination of the SST turbulence model and the transitional model developed by Menter at el.[7] to regard the different flow regimes (laminar and turbulent flow) 2D (flat plate and a NACA profile 66-009) and 3D geometries (NACA wing 16020 and a propeller) were investigated. Painting tests were carried out to validate the numerical calculation results.

Study of scaling problems requires highly accurate numerical simulations in model and full-scale. An approach was developed to ensure the high comparability of the numerical results. Based on the results of the numerical simulations a extrapolation method was developed, which considers both the main geometrical parameters of the propeller geometry and the radial distribution of the force differences between full-scale and model.

Keywords

Propeller, scale effect, viscous flow, laminar and turbulent flow transition, extrapolation method

1 INTRODUCTION

The optimization of the propulsion system becomes important. Through improvements of the hull shape and geometry of the propeller an increase of the propulsive efficiency and a decrease of the operating cost can be achieved.

Due to the rapid increase of ship size and ship speed particularly in the case of container vessels, the diameter of the appropriate propeller becomes larger. Since the dimensions of the test facilities in the towing tanks are not expanded to consider these ship sizes, the used global scale factors increase. With the increase of the scale factors, the significance of extrapolation methods of the measured model results to the full-scale is intensified substantially. The available scaling procedures are applied in a new range, for which only limited experience exists.

The numerical method for the calculation of the viscous flow is a good basis for the investigation of the flow characteristics of propellers in model and full-scale. They open new perspectives and possibilities to gain extensive information about the characteristics of the propeller flow, for example the distribution of the wall-shear stress on the propeller blade. This information is very important for the correct prediction of the propulsion characteristics of a new design. Such information cannot be generated with potential flow-theory based methods.

For the prediction of full-scale design mostly experimentally determined thrust and torque data of the propeller are used. This presupposes that the measuring data are extrapolated as exactly as possible from model to full-scale.

The procedure of carrying out of open water tests in towing tanks is highly optimized. As a consequence the costs and time required for the experimental investigation of the propeller characteristics are clearly minimized. The improvement in the measuring technique makes it possible that the accuracy of the measured data and the reliability of the measuring systems achieve a high level.

In contrast the method to assign the results of the measurement to the full-scale design was not paid the same attention. At present the extrapolation of thrust and torque coefficient takes place in the towing tanks either based on the ITTC recommended procedure [1] or by use of the method of Meyne [2].

The standard ITTC 1978 procedure for performance prediction uses two global corrections, one for the thrust and one for the torque coefficient. These corrections consider the influence of the Reynolds number (Rn), the profile thickness-ratio (t/c) and the pitch ratio (P/D). Thus, the ITTC procedure is hardly able to consider the local flow conditions. That is also not the aim of the ITTC-procedure, which has to be oriented towards practical solutions for reasons of costs.

The procedure developed by Meyne based on the method of the equivalent profile of Lerbs. This requires that the examined propeller is to be regarded as moderately loaded, that means that the propeller possesses an elliptical circulation distribution. Moreover the method is based on the assumption that the lift coefficient depends only on the flow direction and the profile shape, not on the Reynolds number. Therefore the lift coefficients for model and full-scale design are equal.

The relevance of the problem has been recognized by various scientists who applied numerical methods to investigate the characteristics of the flow at model and full-scale. Abdel Maksoud and Heinke [3] examined scale effects of ducted propellers, Müller and Abdel Maksoud [4, 5] as well as Li, Berchiche and Janson [6] accomplished investigations for conventional propellers.

2 APPROACH FOR THE DEVELOPMENT OF THE SCALING METHOD

The aim of the study was the development of an extrapolation method which uses detailed information on the flow behaviour in model and full-scale. The consideration of these effects during the design process permits an improved full-scale propeller design.

For the investigation of the scale effects, 23 propellers representing propellers for large container ships were selected; all of them were developed by MMG. For the determination of the scale effects the flow on these propellers in model and full-scale was simulated numerically. The basis for the development of the scaling method was identified by comparing the computed distribution of the pressure and the wall shear stress on model to the full-scale propeller geometries.

Open water model tests of propellers are conducted in the Reynolds number range, where the transition from laminar to turbulent flow takes place. For the simulation of the flow conditions within this range different numerical calculations were carried out. These investigations contain the flow on a flat plate, NACA profile 66-009 (2D), NACA wing 16020 (3D) and a propeller. For the validation of the numerical computation method used, particularly for the question of the transition from laminar to turbulent flow, painting tests were carried out for open water model tests conditions in the Hamburg Ship Model Basin (HSVA).

For the numerical investigation of the viscous flow on the different geometries the software package Ansys CFX-10 was used. The procedure is a commercial RANSE solver which is sufficiently validated for the computation of propeller flow as well as thrust and moment coefficients.

For the consideration of the turbulence the SST model was applied. For the investigation of the transition of the flow from laminar to turbulent in model scale the transition model [7] was additionally employed.

3 PRELIMINARY INVESTIGATION

In the computation of the flow on the flat plate three different flow regimes were considered: laminar, turbulent and partially-turbulent (transition method).

The characteristics of the flow and therefore the fraction of laminar and turbulent flow on a surface can be documented with the help of the analysis of the wall shear stress. Friction lines show the variation of the friction coefficient (c_F) over the Reynolds number (figure 1).

The results gained by use of the transition model can be described as follows: At low Reynolds numbers the calculated frictional coefficient shows a similar behaviour as the Blasius-line. At $Rn=1.5x10^6$ the transition from laminar to turbulent characteristics takes place. At first this happens abruptly and in the further process the curve of the results of partially-turbulent flow follows the behaviour of the turbulent computed solution, which corresponds to the Prandtl or von Karman-line.

The flow on the NACA profile 66-009 (2D) was computed by using the transition model for different angles of attack. The flow on the symmetrical profile was additionally calculated as fully turbulent. Turbulence in flows is a result of the velocity gradients. They arise on the surface of bodies due to the no-slip condition. The angle of attack of the profile generates additional velocity and pressure gradients, which lead to an earlier transition of the flow. Figure 2 supports this conclusion, the transition of the flow moves to a lower Reynolds number if the angle of attack is increased.

For a detailed analysis of the flow characteristic nearby the surface of the profile the turbulence intensity and the shear stress based on friction were examined. The sudden rise of these values is caused by the transition of the flow from laminar to turbulent. The local position of the transition depends on of the Reynolds number (flow velocity), the angle of attack and the regarded side of the profile.

The data of the wall shear stress was evaluated on the surface of geometry. The values were normalized with the stagnation pressure and represented over the chord length, see figure 3. Parallel to the analysis of the wall shear stress the consideration of the turbulence intensity gives additional valuable information. The position on the profile where the wall shear stress increases corresponds with the rise of the turbulence intensity.

The NACA wing 16020 has an elliptic radial distribution of the chord length: The wing is fixed at one side; the other is free like a propeller tip. The wing is investigated at different scales. The smallest one has a span of 0.125m (relates to a typical model propeller radius). The flow on the wing was calculated using the transition model. At full scale the flow is considered as fully turbulent, since the laminar fraction can be considered to be negligible. The analysis of the pressure distribution on all investigated Reynolds numbers at the same section (r/R) shows only small differences near the hub region. The differences in pressure between the smallest and the largest investigated wing clearly increase towards the tip. The pressure difference between model and full-scale is a function of the radial position. Therefore the problem of the scale effects cannot be reduced to a 2D-problem.

4 DETERMINATION OF THE SCALE EFFECTS OF DIFFERENT PROPELLER GEOMETRIES

4.1 Geometrical characteristics of the investigated propellers

A new extrapolation method should be able to take into account the main propeller parameters which can influence the scale effects, for example number of blades (z), propeller pitch ratio (P/D), expended area ratio (EAR), skew angle (θ). For this reason 23 typical propellers for large container ships were selected. The geometrical characteristics of the propellers are listed in table 1.

Table 1: Geometrical characteristics of the investigated propellers

z	$D_{full\text{-}scale}$ [m]	P/D	EAR	Skew [θ°]
4-6	5,15-10,5	0,66-1,07	0,53-1,03	17-45

The Reynolds number based on the chord length at r/R=0.7 extends for the model size between $5.0x10^5$ and $9.5x10^5$, for the full-scale design between $3.5x10^7$ and $1.1x10^8$.

4.2 Creation of the CAD data and the mesh

For the numerical calculation of the viscous flow a very accurate propeller geometry description is required, particularly at the blade tip and at the intersection between blade root and hub. In order to guarantee a completely automated configuration of the propeller surface, special attention was paid to the surface in the tip region, since no sufficient geometrical information was available in the original data. The necessary surfaces for the boundaries of the simulation domain were also to be created. The construction of the interface between the individual blades becomes more difficult with every added blade. Since this position must not influence the computation results, test calculations were carried out to rule out any such possible effects. A sketch of the entire computing domain is shown in figure 4, and figure 5 shows the subsection of the rotor.

For the description of the periodic boundaries (interface between the blades) a surface was created on basis of the pitch and chord length at the individual radius and in each case turned around the angle ± (360/2)/z over the suction and pressure side of the blade.

The diameter of the interfaces of the rotating subsection was defined as a function of the propeller diameter. The dimensions of the simulation domain are: length of the inflow region 10D, length of the region behind the propeller 20D, radius of the grid 5D, radius of rotor domain 1,5D and length of rotor domain 0,5D. The total number of cells is about four millions; half of them are located in the rotor domain and the other half in the stationary domain.

The boundaries of the stationary part of the computing domain are quite simple to create since it is not necessary to keep face to face connection of the control volumes at the interfaces between the rotor domain and the stationary part of the grid. At these faces a general grid interface boundary condition is applied.

A block-structured grid is applied. Special attention was paid to the development of a suitable topology of the mesh. Due to the lack of sufficient mathematical formulations for the surface of the tip region it was not possible to apply an automatic grid generation process. The grid generation was carried out by adjusting an existing mesh on the new propeller geometry. In this way it was possible to ensure a similarity of all meshes. The distribution of the mesh points within the range near the wall must be adapted to the expected boundary layer thickness of the respective Reynolds number. However the sufficient resolution of the computing grid within the range near the wall can only be determined when a solution for the flow field is achieved. Therefore an analysis of the point distribution orthogonal to the wall surface was carried out and, if necessary, an adjustment of the distribution after each computation was done. Open water conditions were accomplished for all computations. Due to the parallel inflow the numerical simulations could be limited to one blade. The interaction between the blades was considered as a spatial periodic boundary condition.

4.3 Numerical investigations

For the simulation of the viscous flow of the propellers in model scale the transition model was used in combination with the SST turbulence model. Since the transition model requires information on the turbulence intensity and this factor is usually not measured by the towing tank laboratories, test calculations with different turbulence intensities (2.5%, 5.0%, 7.5%) were carried out. In the region of the design point a good agreement with the model results could be achieved by use of 5% turbulence intensity at the inlet plane. For the computation of the full-scale propeller the flow is considered fully turbulent, so that the SST turbulence model in the standard form was applied.

The flow on the investigated propellers was simulated at advance ratios ranging from 0.1 to 1.0. The computed thrust and torque coefficients for model scale were compared with the model test results. The largest deviations occur at small advance ratios as well as at advance ratios behind the maximum efficiency. The computations at these advance ratios have a strong unsteady character; therefore the applied steady computation could not be able to capture the important detail of the flow on the propeller blade. Since the extrapolation method should be integrated into the propeller design process, the main focus is laid on the advance ratios close to the operating point of the propeller. This corresponds to the operation range of the ships.

Painting tests were carried out for four model propellers in the HSVA. The results of the tests were qualitatively compared with the results of the numerical computations. Figure 6 shows the wall shear stress as well as the streamlines on the surface of the suction side of the blade. The transition from laminar to turbulent flow is shown clearly where a locally strong rise (change from blue to green) of the wall shear stress takes place. For the illustration of the transition region the blades in the painting test were provided with different coatings. The range of the blade in which the colour was washed off marks the turbulent flow range. Moreover the streamlines which are not running along a radius section can still be recognized in the remaining colour confirming the laminar flow behaviour (see figure 7). This description is valid for both sides of the blade (figure 8 shows the numerically computed results and figure 9 the painting test for the pressure side).

5 Scaling method

For the development of the formula of the scale effects the differences (full-scale to model) of the resulting forces and its effective direction (angle α) were determined from the axial and tangential force components between model and full-scale at each radius of the blade. Figure 10 shows the forces and the effective direction of the resultant force (angle α) of the propeller model and of the full-scale.

The scale-dependent difference of the radial distribution of the resultant force and the direction (angle) are building the basis for the extrapolation method. Figure 11 shows the radial distribution of the differences (full-scale to model) of the resultant forces and figure 12 the effective force direction. Average values of the resultant force and the angle of attack at several sections of the propeller blade were computed using data from all investigated propeller geometries; the results can be represented by formula as follows:

$$\Delta F_{r/R} = \frac{\sum_{i=1}^{n} \Delta F_{r/R}}{n} + a \cdot Skew + b \cdot \frac{EAR}{z} + c \cdot P/D + d \cdot C_{TH} \qquad (1)$$

$$\Delta \alpha_{r/R} = \frac{\sum_{i=1}^{n} \Delta \alpha_{r/R}}{n} + e \cdot Skew + f \cdot \frac{EAR}{z} + g \cdot P/D + h \cdot C_{TH} \qquad (2)$$

In addition, part of the correction functions are dependent on the geometrical characteristics (P/D, EAR/Z, Skew) as well as the thrust loading. The coefficients were computed by applying the least squares method.

The employment of the above developed formula for computation of the full-scale thrust and torque coefficients requires data from the design procedure of the propeller as well as from the model tests. The magnitude and direction of the resultant forces at each radius of the propeller blade, as well as thrust and torque are needed from the propeller design procedure. From the measuring data the measured total thrust and torque of the propeller and the test conditions are used.

The computations are carried out for each radius of the blade as follows:

- With help of the developed formula the average changes of the total force and the angle are determined, that is the first term on the right side of formula 1 and 2.
- The corrections are determined according to the geometrical characteristics of the propeller by using the other terms on the right side of the formulas
- The newly corrected forces are divided with consideration of the modified angle into an axial and a tangential component and the new thrust and torque coefficients for each radius can be calculated.

Subsequently, an integration of the thrust and torque coefficients over the radius delivers the thrust and torque coefficients for the full-scale propeller.

The measured results for thrust and torque coefficients are used to scale the calculated radial distribution of the forces in model scale. That is necessary to overcome any inaccuracies in the design method caused by estimation of the forces on the propeller blade.

Figure 13 shows the result for one of the investigated propellers in model scale. The thrust and torque coefficients for the propeller in full–scale were estimated with help from the extrapolation methods of the ITTC, Meyne and the new method developed within this study.

6 Conclusion

The viscose flow around a high number of propellers was investigated in the model (Rn between $5{,}0 \times 10^5$ and $9{,}5 \times 10^5$) and full-scale design (Rn between $3{,}5 \times 10^7$ and $1{,}1 \times 10^8$). It was considered that in the model scale the flow is not fully turbulent. The numerical results show that three-dimensional effects play an important role in the topic of scale effects and a reduction of the problem on a two-dimensional case (profiles) is not sufficient to capture all aspects of the different behaviour.

The developed scaling procedure supplies the change of the magnitude of the resultant force and their angle at each radius of the propeller blade. Since the force difference between model and full-scale design is a function of the nondimensional propeller radius, the developed procedure can provide detailed information regarding the scale effects. It considers the global geometrical parameters of the propeller and the threedimensionality of the flow so far as the influences of the geometrical characteristics are covered by the investigated propeller geometries.

ACKNOWLEDGEMENT

The authors thank the German Federal Ministry for Economics and Technology as well as the project management organization Jülich (department for navigation and marine technology) for supporting the project.

Many thanks also to Dr.-Ing. habil. K. Wagner and Dr.-Ing. K. Meyne for their valuable scientific contributions, as well as the HSVA for the free supply of the middle cavitation tunnel to realize painting tests of propellers. We particularly thank Mr. Dipl. -Ing. O. Zarbock and the students at the Institute of Ship Technology and Transport Systems who work on the HyProCon project for their great engagement in the project.

REFERENCES

1. SNAME. (1988) ‘Principles of Naval Architecture – Volume II – Resistance, Propulsion and Vibration’. SNAME, USA.
2. Meyne, K. (1972). ‘Untersuchung der Propellergrenzschichtströmung und der Einfluss der Reibung auf die Propellerkenngrößen’, STG-Jahrbuch
3. Abdel-Maksoud, M., Heinke: H.-J. (2002). ‘Scale effects on Ducted Propellers’. 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan
4. Müller, S.-B, Abdel-Maksoud, M. (2005). ‘Influence of Scale Effects on the Characteristics of Propellers, CFD 2005’. CFD Technology in Ship Hydrodynamics, Southampton, England.
5. Müller, S.-B, Abdel-Maksoud, M. (2005). ‘Einfluss der Reynoldszahl auf die Kennwerte von Propellern’. 100. Hauptversammlung der Schiffbautechnischen Gesellschaft (STG), Berlin, Germany.
6. Li, D.-Q., Berchiche, N., Janson, C.-E. (2006). ‘Influence of Turbulence Models on the Prediction of Full-Scale Propeller Open Water Characteristics with RANS methods’. 26th Symposium on Naval Hydrodynamics, Rome, Italy.
7. Menter, F. R., Langtry, R., Völker, S. (2006) ‘Transition Modelling for General Purpose CFD Codes’. Flow, Turbulence and Combustion, Volume 77, Numbers 1-4, Springer Netherlands

FIGURES

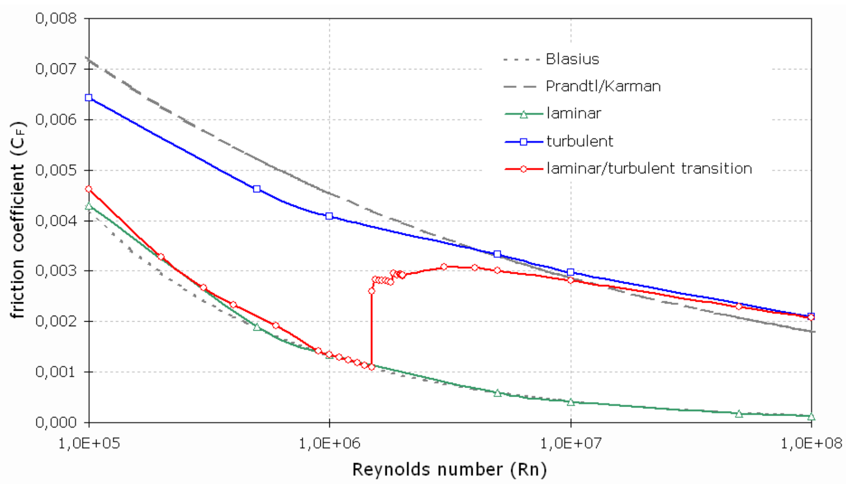

Figure 1: Flat plate – change of the friction coefficient c_F as a function of Rn

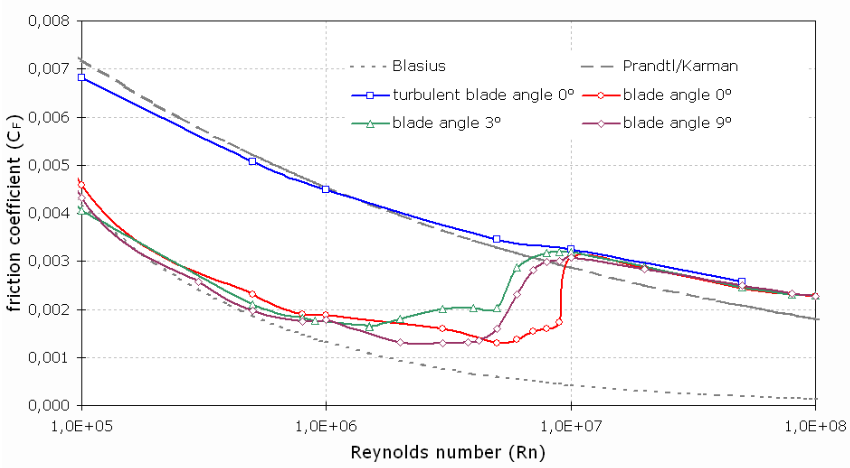

Figure 2: NACA 66-009 profile - change of the friction coefficient c_F as a function of Rn

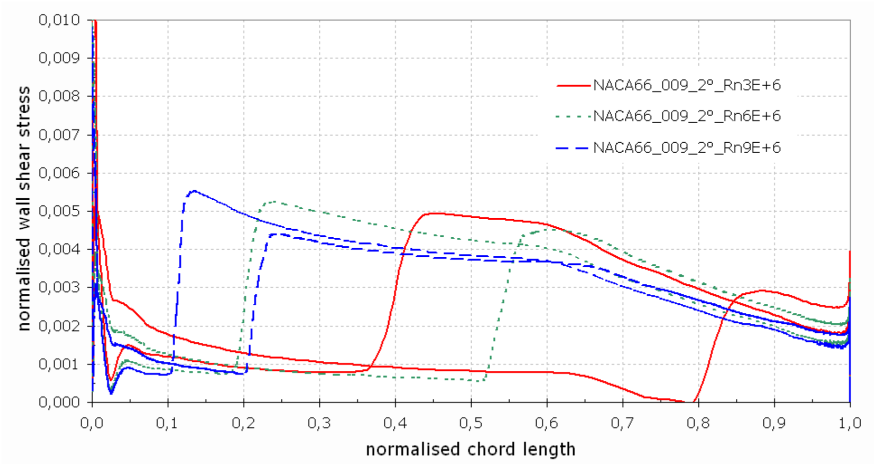

Figure 3: NACA 66-009 profile – normalised wall shear stress at angle of attack 2°

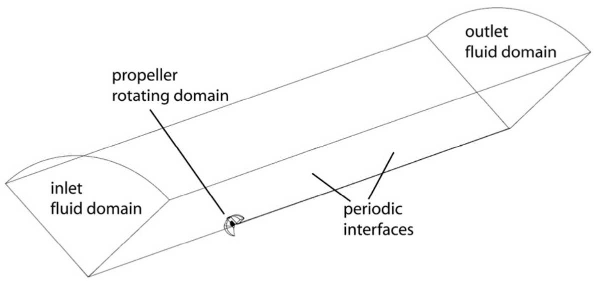

Figure 4: Sketch of the simulated area and the used boundary conditions

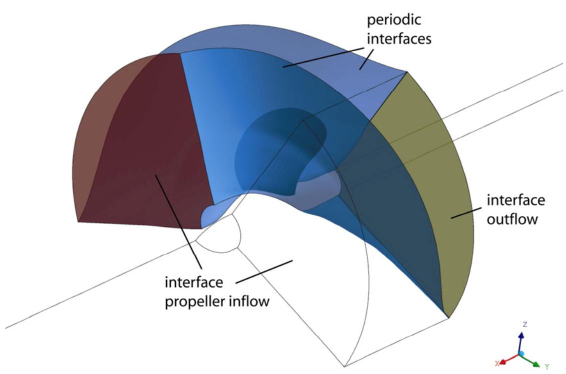

Figure 5: Rotor domain

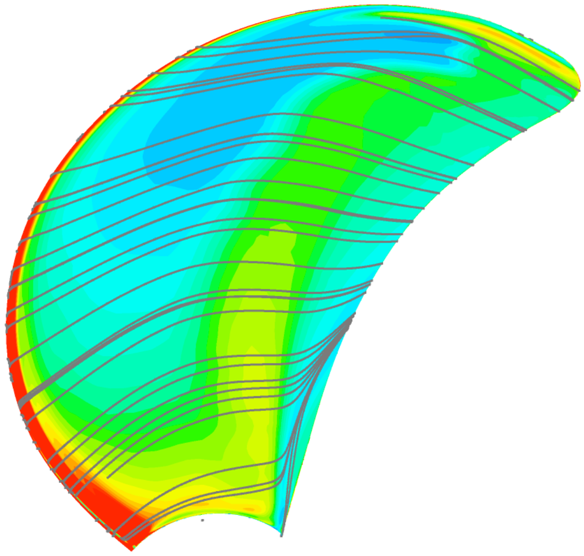

Figure 6: Suction side - wall shear stress and streamlines

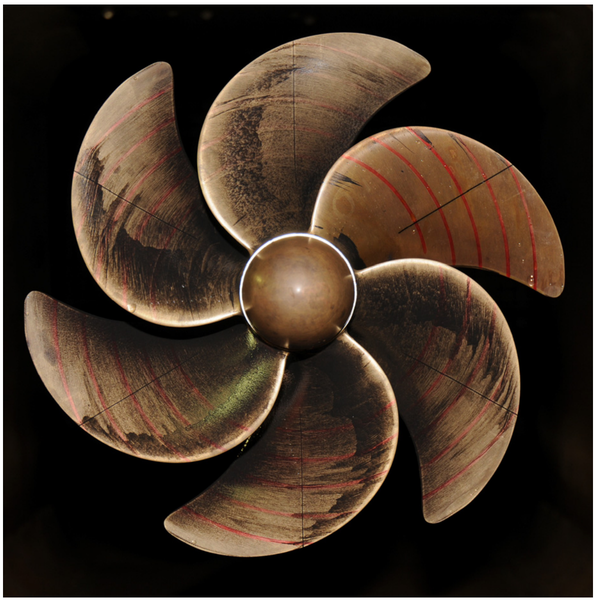

Figure 7: Suction side – painting test

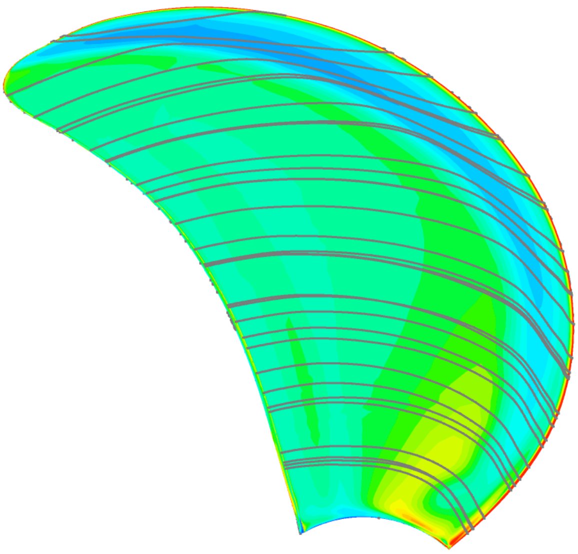

Figure 8: Pressure side - wall shear stress and streamlines

Figure 9: Pressure side – painting test

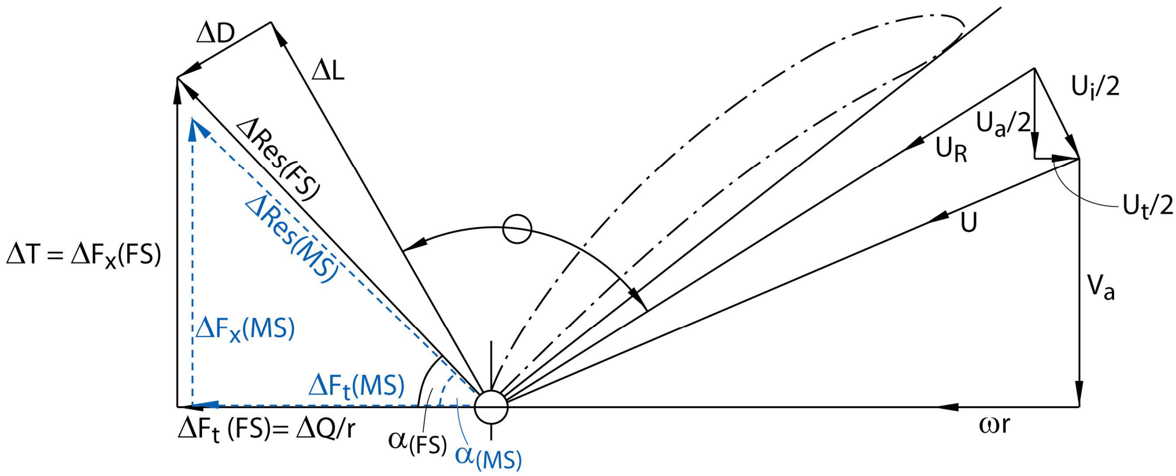

Figure 10: Forces on the sections of the propeller blade and its effective direction

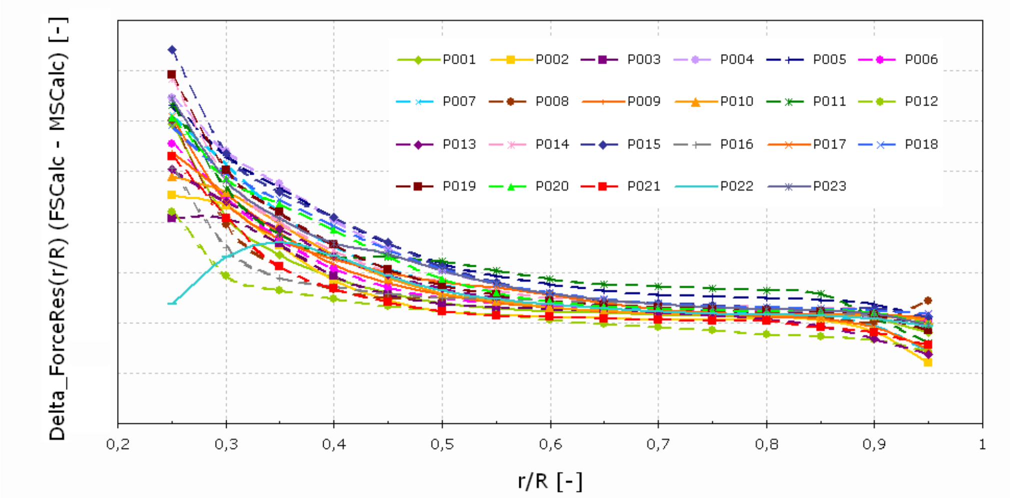

Figure 11: Differences of the resultant force (full-scale - model)

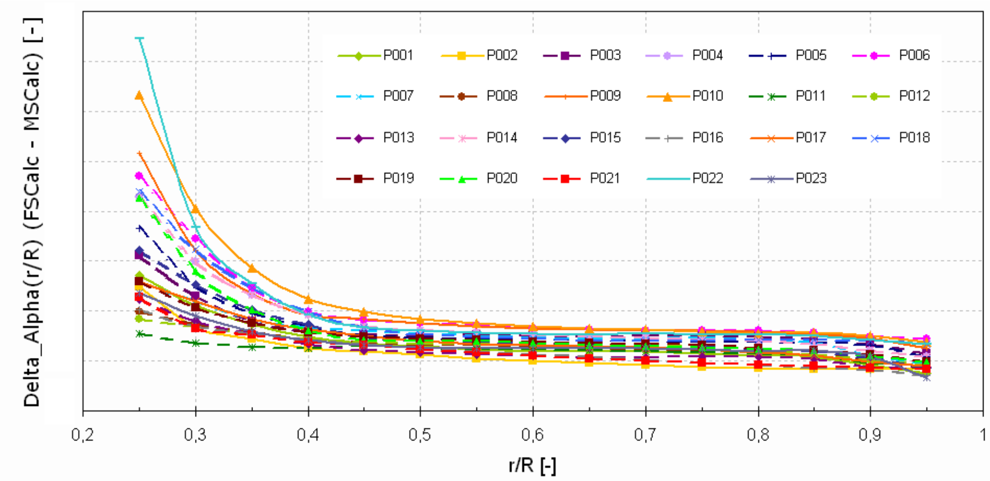

Figure 11: Differences of the effective direction (full-scale – model)

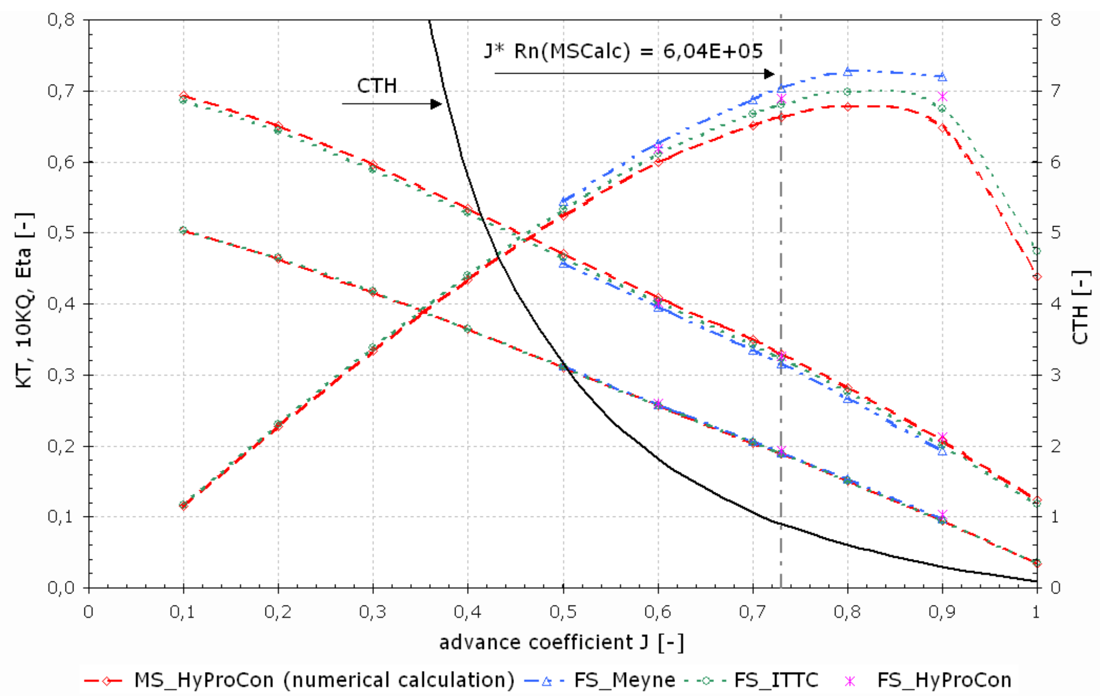

Figure 13: Open water diagram for model and full scale