THESIS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY IN THERMO AND FLUID DYNAMICS

Marine Propulsion System Performance Beyond the Propulsive Factors

JENNIE ANDERSSON

Department of Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2021

Marine Propulsion System Performance Beyond the Propulsive Factors
JENNIE ANDERSSON
ISBN 978-91-7905-566-0

Doktorsavhandlingar vid Chalmers tekniska högskola
Ny serie nr. 5033
ISSN 0346-718X
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Chalmers Digitaltryck
Göteborg, Sweden 2021

Marine Propulsion System Performance Beyond the Propulsive Factors

JENNIE ANDERSSON
Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Abstract

The marine propulsion system often operates in the wake of the ship it is mounted on. This implies that the propulsion system affects the flow around the hull, and the inflow to the propulsion system is dependent on the hull shape. Due to the complexity of the flow, the performance of marine propulsion systems has historically been assessed through model-scale testing in basins, alternative numerical tools were for a long time not available. However, the limited possible measurements does not provide any detailed descriptions of the propulsion system performance. A second factor complicating the use of model-scale testing is the inevitable low Reynolds number in relation to the real ship. The need for a more detailed description of the propulsion system performance motivates the use of Computational Fluid Dynamics (CFD); there are no limitations on what to extract in terms of flow details or forces on surfaces and it can be applied for the ship-scale system. However, this requires a representative model of the flow in ship-scale which is not yet well established. In this thesis, using CFD, two alternative approaches to the propulsive factors originating from the model-test scaling procedure are proposed to describe the propulsion system interaction effects: A detailed evaluation of forces on the propulsion system and hull surfaces and a control volume approach based on energy fluxes describing the power required by the propulsion system in terms of various energy fluxes. For the first approach a powerful tool is the thrust over torque coefficient (K_T/K_Q) for a blade around a revolution, or studies of its radial distribution at specific positions. A clear advantage of the control volume approach is its possibilities to describe the viscous losses. As a step towards ship-scale CFD a review and comparison of different methods to model hull roughness is conducted; it shows no convergence towards specific roughness functions or methods to obtain the roughness length scales and there is neither a strong correlation between the additional resistance predicted by various hull roughness models and the Average Hull Roughness (AHR). Applying the proposed approaches, a few generic interaction effects could be explained. For instance: The old rule of thumb regarding optimal propeller diameter in-behind based on model-scale tests from the 1950s is shown to be mainly due to that operation together with a rudder favours smaller propeller diameters since the rudder can make use of the stronger slipstream rotation. However, the results indicate that this only holds within the same scale factor, and the optimal propeller diameter for the ship is most probably larger than what is indicated by propeller series data. Other generic interaction effects explained is how tip-unloading deteriorates propeller performance to a larger extent in-behind since the wake distribution further decreases the load on the blade tip, and how blunter leading edges has a superior performance at low load, since they are less sensitive to poor performance at negative angles of attack.

Keywords: Propulsor-hull interaction, Energy balance analysis, CFD, RANS

List of Publications

This thesis consists of an extended summary and the appended papers listed below. A summary of each paper including division of work is available in Section 4.

Paper I J. Andersson, A. Eslamdoost, A. Capitao-Patrao, M. Hyensjö, and R. E. Bensow. Energy Balance Analysis of a Propeller in Open Water. *Ocean Engineering* **158** (2018), 162–170

Paper II J. Andersson, A. Eslamdoost, M. Vikström, and R. E. Bensow. Energy balance analysis of model-scale vessel with open and ducted propeller configuration. *Ocean Engineering* **167** (2018), 369–379

Paper III J. Andersson, D. R. Oliveira, I. Yeginbayeva, M. Leer-Andersen, and R. E. Bensow. Review and comparison of methods to model ship hull roughness. *Applied Ocean Research* **99** (2020)

Paper IV J. Andersson, R. Gustafsson, A. Eslamdoost, and R. E. Bensow. On the Selection of Optimal Propeller Diameter for a 120-m Cargo Vessel. *Journal of Ship Research* **65**.2 (2021), 153–166

Paper V J. Andersson, R. Gustafsson, R. Johansson, and R. E. Bensow. Propeller-Hull Interaction Beyond the Propulsive Factors. *Ocean Engineering (under review)* ()

In addition to the appended papers, I have authored or co-authored the following publications/reports:

Paper A J. Andersson, M. Hyensjö, A. Eslamdoost, and R. E. Bensow. "CFD Simulations of the Japan Bulk Carrier Test Case". *Proceedings of the 18th Numerical Towing Tank Symposium.* Cortona, Italy, 2015

Paper B J. Andersson. *CFR 137: Analyzing Ship Propulsion Interaction Effects - Review of Published Methods.* Internal RR-report, 2015

Paper C A. Eslamdoost, J. Andersson, R. Bensow, R. Gustafsson, and M. Hyensjö. "Analysis of Propeller-Hull Interaction Phenomena on a Self-Propelled Axisymmetric Body". *Proceedings of the Fifth International Symposium on Marine Propulsors.* Espoo, Finland, 2017

Paper D J. Andersson, R. Gustafsson, A. Eslamdoost, and R. E. Bensow. "On the Selection of Optimal Propeller Diameter for a 120m Cargo Vessel". *SNAME Propeller-Shafting 2018 Symposium.* Norfolk, Virginia, USA, 2018

Paper E J. Andersson. *CFR 168: Turbulence Model Evaluation Using the KVLCC2 Test Case.* Internal RR-report, 2019

Paper F A. Eslamdoost, J. Andersson, R. Bensow, and M. Vikström. "Application of Control Volume Energy Balance for Analysing Propeller-Hull Interaction in Presence of Free-surface". *Proceedings of the Sixth International Symposium on Marine Propulsors.* Rome, Italy, 2019

Paper G J. Andersson, R. Gustafsson, R. Johansson, and R. E. Bensow. "In-Behind Performance of an Ice Classed Propeller in Model and Ship-Scale". *Proceedings of IX International Conference on Computational Methods in Marine Engineering, MARINE 2021.* Online, 2021

Paper H J. Andersson, A. Abolfazl Shiri, R. E. Bensow, J. Yixing, W. Chengsheng, Q. Gengyao, G. Deng, P. Queutey, Y. Xing-Kaeding, P. Horn, T. Lucke, H. Kobayashi, K. Ohashi, N. Sakamoto, F. Yang, Y. Gao, B. Windén, M. G. Meyerson, K. J. Maki, S. Turnock, D. Hudson, J. Banks, M. Terziev, T. Tezdogan, F. Vesting, T. Hino, and S. Werner. Ship-Scale CFD Benchmark Study of a Pre-Swirl Duct on KVLCC2. *Applied Ocean Research (under review)* ()

Acknowledgements

This work had not been possible without all the great people that have surrounded me and provided advice, feedback, and questioned the work. First I would like to express my deepest gratitude to my main supervisor Rickard E. Bensow for his support during the project. I am very thankful for the experience, both considering technical and pedagogical aspects, that he has brought into the project. In addition I greatly acknowledge the freedom he has been giving me to explore my ideas. I would also like to thank my supervisors and colleagues at Kongsberg Maritime, previously Rolls-Royce, both in Kristinehamn and Ålesund/Ulsteinvik. Without the relevant problems you have provided me with, as well as industrial knowledge, this work would not have been possible. Especially I would like to thank Robert Gustafsson, Marko Vikström, Rikard Johansson, Kåre Nerland, Göran Grunditz, Johan Lundberg, and Geir-Åge Øye for engaging in my project. I would also like to acknowledge my colleagues at the department of Mechanics and Maritime sciences, for providing an inspiring working environment. My co-supervisor Arash Eslamdoost for careful proofreading throughout, Muye Ge for excellent cooperation in all matters I ever turned to you with, Fabian Thies for his never-ending support as a highly competent Naval Architect, Alexandre Capitao-Patrao for fruitful discussions concerning energy flux balance analyses and Daniel Lindblad for his curiosity with regards to my project. Finally, I would like to thank my patient husband Marcus for all support and tolerance with my ups and downs, as well as our sons Isak and Arvid, for their inspiring and never-ending positive view of life.

This research was mainly supported by the Swedish Energy Agency (grant number 38849-1 and 38849-2) and Kongsberg Maritime Sweden AB through the University Technology Centre in Computational Hydrodynamics hosted by the Department of Mechanics and Maritime Sciences at Chalmers. Financial support has also been received through the Swedish Transport Administration (grant number TRV-2018/76560 and TRV-2018/76544). All financial support is gratefully acknowledged. The computer resources at Chalmers Centre for Computational Science and Engineering (C3SE) and at the National Supercomputing Centre (NSC) in Linköping, both provided by the Swedish National Infrastructure for Computing (SNIC), are also gratefully acknowledged.

Nomenclature

c_p	Specific heat capacity
E	Energy
F_D	Tow force applied in self-propulsion test
e	Energy per unit mass
D	Propeller diameter
J	Advance ratio
K_Q	Torque coefficient, $Q/(\rho n^2 D^5)$
K_T	Thrust coefficient, $T/(\rho n^2 D^4)$
k	Turbulent kinetic energy
k_s	Equivalent sand grain roughness
n	Rotation rate
$\overrightarrow{n}$	Normal unit vector
P_D	Delivered power
P_E	Effective power
p	Pressure
$\dot{Q}$	Heat transfer rate
Q	Torque
R	Resistance
T	Thrust
t	Thrust deduction factor
T	Temperature
$\hat{u}$	Internal energy
$\overrightarrow{V}$	Velocity vector
V_A	Advance velocity
V_S	Ship velocity
V_r	Radial velocity component
V_t	Tangential velocity component
V_x	Axial velocity component
$\dot{W}$	Rate at which work is done by the system
w_T	Taylor wake fraction

Greek symbols

η_D	Propulsive efficiency
η_H	Hull efficiency
η_O	Open water efficiency
η_R	Relative rotative efficiency
ρ	Density
$\vec{\tau}$	Shear stress vector

Abbreviations

AHR	Average hull roughness
CFD	Computational Fluid Dynamics
CS	Control surface
CV	Control volume
EASM	Explicit algebraic stress model
ITTC	International Towing Tank Conference
MRF	Multiple reference frames
RANS	Reynolds Averaged Navier-Stokes
RSM	Reynolds stress model

Contents

1 Introduction

The strive towards more fuel efficient ships is a continuously ongoing process, motivated by both economic and regulatory reasons. The regulatory drive stems from the target of the International Maritime Organization (IMO) to reduce the total annual greenhouse gas emissions from international shipping by at least 50% by 2050 compared to 2008 [14]. In the hydrodynamic design of a ship, important aspects to consider for the final fuel consumption are both ship resistance and the marine propulsion system performance, both in relevant operating conditions.

A marine propulsion system is often operating in the wake or boundary layer of the ship it is mounted on. This implies that the propulsion system affects the flow around the ship, and the inflow to the propulsion system is dependent on the hull shape of the ship. This complete system, the propulsion unit, and possibly other appendages such as rudder and energy saving devices (ESDs), operating together with the ship, is what will be referred to as the marine propulsion system. It is a complex system, where it is not meaningful to describe the performance of each component isolated, they rather must be treated as parts of a larger system.

Detailed knowledge on marine propulsion system performance, and how the different components interact with each other under a variety of operating conditions, is generally not as widespread. This is mainly a consequence of the complexity of the system, but also considered to be influenced by factors such as market structure, where different corporations are often delivering the separate components within the propulsion system. Another factor influencing the general system understanding is the limitations of model-scale testing.

1.1 Model-Scale Testing and the Propulsive Factors

Model-scale testing in basins has historically been the main tool for verification and description of marine propulsion system performance. Due to the complexity of the flow, with a propeller operating in a turbulent wake, large variations in operating conditions during a revolution, and possible interaction with the free surface, alternative numerical tools has for a long time not been available. The methodology commonly applied for power prediction based on model-scale testing is outlined in the International Towing Tank Conference (ITTC) recommended procedure for performance prediction [15], and includes three separate tests as illustrated in Figure 1.1: the propeller open-water test with the propeller operating in homogeneous inflow to establish its performance in relation to loading; the resistance test, i.e. the hull towed in water without propeller to measure its resistance; and the self-propulsion test where the propeller is mounted on the hull and it operates as a complete system.

The resistance and self-propulsion tests are conducted at Froude number similarity to keep the relationship between flow inertia and gravity constant, i.e. the surface wave

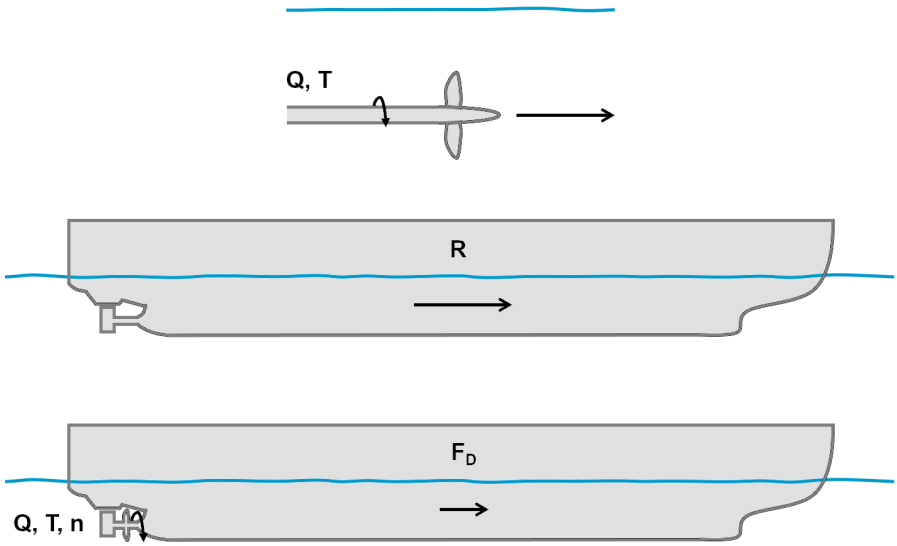

Figure 1.1: *Illustration of tests conducted for power prediction according to the ITTC-78 performance prediction method [15]. Open water test (top), resistance test (middle), and self-propulsion test (bottom).*

pattern scales correctly between the ship and the model. When Froude number similarity is met it implies that Reynolds number (Re) similarity cannot be met, at least not for any practically possible test arrangements. The Reynolds number describes the relationship between inertial and viscous forces for a fluid, which hence will be different between the model-scale setup and the ship. The model will operate at significantly lower Reynolds numbers, around 10^6, compared to the ship, about $10^8 - 10^9$, i.e. the viscous forces will be of higher importance for the model. Direct consequences of this are thicker hull boundary layers and larger wake, i.e. the propulsion system operating conditions may vary significantly between ship and model-scale setup.

Another critical impact by the low Reynolds number operation is the influence on boundary layer formation. On the hull, various measures to ensure laminar to turbulent transition downstream the bow are commonly applied, however this is most often not conducted for the propeller and other propulsion system components. For the ship these boundary layers can generally be assumed to be fully turbulent, but for the model the risk for partly laminar boundary layers is obvious. The general turbulence level in the wake is high and facilitates transition to turbulent boundary layers, although the complete propulsion system may not operate in such high turbulence levels so that laminar boundary layers completely can be avoided. The laminar to turbulent transition may also be influenced by geometrical factors of the propulsion system, which implies that a comparison of two systems in model-scale may be influenced by their respective sensitivity to boundary layer transition. The low Reynolds number of each component in the propulsion system also implies thicker boundary layers and an increased risk of boundary

layer separation, in relation to ship-scale. The sensitivity to boundary layer separation may also differ between geometries, i.e. yet an additional geometrical factor that may be of importance in model-scale but not for the real system.

To obtain the final power prediction for a ship, considering the Reynolds number differences between the model and the ship, the scaling procedure as outlined in the ITTC-78 performance prediction method [15] is commonly applied. For the resistance test the inability to match the ship Reynolds number implies that the hull resistance would be overestimated if the model-scale value was scaled directly using force coefficients. Instead, to scale the measured hull resistance to ship-scale, it is separated into a residuary and a frictional part, were the second is obtained and scaled applying a flat plate analogy [16]. The scaled hull resistance is also compensated by a number of factors to account for roughness effects, air drag etc. In the self-propulsion test, to compensate for the low Reynolds number, and hence high share of viscous resistance, the propeller is partly unloaded by a towing force F_D applied to the hull, as indicated in Figure 1.1. The final predicted power is obtained through,

$$P_D = \frac{P_E}{\eta_D}, \tag{1.1}$$

where P_E is the effective power, i.e. the scaled resistance of the ship without working propeller multiplied with the ship speed, and η_D is the propulsive efficiency which is defined as:

$$\eta_D = \eta_O \eta_R \eta_H. \tag{1.2}$$

Here, η_O is the open water efficiency obtained from the open water test, evaluated applying K_T-identity, i.e. the operating conditions in open water corresponding to the same thrust coefficient, K_T, as in self-propulsion, and scaled to estimated ship-scale performance. η_R is the relative rotative efficiency, defined as

$$\eta_R = \frac{K_{QO}}{K_Q}. \tag{1.3}$$

It relates the torque coefficient, K_Q, required in open water (index O) evaluated applying K_T-identity to the K_Q required in-behind. η_R is normally considered to be equal in model-scale and for the ship. η_H is the hull efficiency which is formulated as,

$$\eta_H = \frac{1-t}{1-w_T}, \tag{1.4}$$

where t is the thrust deduction factor and w_T the Taylor wake fraction. The thrust deduction factor, t, is defined as

$$t = \frac{T-(R-F_D)}{T} = 1 - \frac{R-F_D}{T}, \tag{1.5}$$

which implies that it is representing the increment in resistance of the hull due to the propeller action, it is commonly assumed to be the same in model-scale and for the ship.

The Taylor wake fraction, w_T, is defined as

$$w_T = \frac{V_S - V_{AT}}{V_S} = 1 - \frac{J_T D n}{V_S}, \tag{1.6}$$

which aims to describe the wake velocity relevant for the propeller in relation to the ship speed. The advance velocity and advance ratio are evaluated applying K_T-identity, represented by the index T. w_T is scaled according to assumed wake differences between model-scale and the ship.

The relative rotative efficiency η_R, the hull efficiency η_H, the thrust deduction factor t, and the Taylor wake fraction w_T, are commonly referred to as the propulsive factors. As described above, the propulsive factors are critical for the ship power prediction (Eq. 1.1) and they are also the only measures to some extent describing the propulsion system performance based on the model-scale tests. Despite this the propulsive factors are amongst many practitioners considered difficult to grasp, which is not strange considering that a few overall variables should cover several complex flow phenomena, including unexpected flow separation and laminar boundary layers caused by the low Reynolds number operation, often not visible through measurements.

In addition to the Reynolds number differences between the model and the ship, there is one more scale-effect complicating the propulsive factors, especially η_R: the Reynolds number differences between the model-scale open water and self-propulsion tests. The propeller open water test is most commonly conducted at as high Reynolds number as possible, i.e. higher than in the self-propulsion test, to minimize unwanted influences of partly laminar boundary layers on the blades. This is motivated by that the risk for laminar boundary layers is larger in the undisturbed inflow of a propeller in open water in relation to in-behind operation in a turbulent wake. Still laminar boundary layers on the propeller blades may be present in both open water and self-propulsion conditions. Differences in performance due to differences in pressure distribution and/or laminar to turbulent transition of boundary layers between open water and self-propulsion tests will enter as differences in K_Q and directly influence η_R, this is illustrated in Paper V. Paint streak tests can be conducted on the propeller blades in open water to identify the extent of the laminar boundary layers and the laminar to turbulent transition line, see for instance [17, 18]. However, paint streaks are more difficult to apply on the propeller blades in self-propulsion due to the strong tangential variation in the flow field [19].

An additional reason to why the propulsive factors are considered difficult to grasp may be that they do not always fully represent what they commonly are considered to describe. For instance may the difference in open water efficiency η_O when comparing two systems be expected to only describe the propellers performance differences in homogeneous inflow. Less attention may be paid to that η_O is evaluated applying K_T-identity, implying that differences in efficiency due to K_T differences also are included, i.e. a large share of system performance is included also in this measure. A practical example of this is provided in Paper V. Another example is the Taylor wake fraction, which commonly is considered to represent the undisturbed velocity in which the propeller operates behind the ship. This is true to some extent but it also includes the suction of the propeller. However, w_T is evaluated using K_T-identity as described in Eq. 1.6 which actually implies that also the system performance is included in the evaluation. A practical example

of this is also provided in Paper V: two different propellers with the same diameter and radial load distribution, mounted on the same hull and conducting the same work, shows a variation in w_T. This would not be expected if w_T only described the undisturbed velocity at the propeller plane. The difference can be deduced from differences in rotation rate, n, due to variations in propulsion system performance.

The necessity for a more detailed description of the propulsion system performance than the propulsive factors has been acknowledged prior to the common use of Computational Fluid Dynamics (CFD) based on the Reynolds Averaged Navier-Stokes (RANS) equations within ship design. Dyne [20] derived a method based on potential flow assumptions, suggesting a propulsive efficiency based on wake losses and gains. The system of propulsive factors was neither developed to provide a deeper understanding of the marine propulsion system performance, but as an engineering tool to scale up experimental data from model-scale. But unfortunately, since favorable propulsive factors in model-scale are important for the power prediction, and therefore also for comparisons between suppliers and contractual agreements, these figures may take the focus from more deeper understanding of the real ship propulsion system performance.

1.2 CFD in Marine Propulsion System Design

During the last decades it has become feasible to complement model-scale tests with various numerical methods, which therefore have increased in both maturity and popularity as design tools for marine propulsion systems. CFD opens up new possibilities to characterize the interaction effects and enhance the understanding of marine propulsion system performance, since there are no limitations on what to extract in terms of flow details or forces on different components. Although, CFD is still only a model of the real flow conditions and the representativeness of that model is to a various extent often unknown.

Validation of CFD models within ship design has for a long time mostly been limited to model-scale validation, due to the lack of measurements on ships. The possibilities to validate modelling of the propulsion system is limited to available experimental data from the self-propulsion test, which often is average torque and thrust for the propeller, sometimes complemented with velocity field measurements at certain instances. Taking into account the complexity of the flow around the marine propulsion system in model-scale, often suffering from both separating regions and partly laminar boundary layers on various components, the measured data may be regarded insufficient.

Considering the influences on propulsion system performance from Reynolds number differences between model and ship, it is promising to note that ship-scale CFD and associated validation is getting more attention. A workshop on ship-scale hydrodynamics was held in 2016 [21]. Further ship-scale validation data has been published, including flow measurements, by for instance Inukai et al. [22], Sakamoto et al. [23], and Wakabayashi et al. [24]. Currently an industry wide research project is ongoing to provide more ship-scale data possible to apply for CFD validation [25]. Further, also the next occasion of the international workshop series on CFD in Ship Hydrodynamics, held since 1980, is planned to include a ship-scale validation case for the first time [26].

There is still a lot of validation work to be carried out for ship-scale CFD models to increase its general maturity and trustworthiness, which is clear from the results in for instance Paper H. However, it is commonly agreed upon that sooner or later numerical methods will replace model-scale testing as the main tool for predicting ship performance in the design phase. As a consequence, the system of propulsive factors will no longer be necessary for scaling purposes, which really opens up for new possibilities to characterise the performance of marine propulsion systems. On the other hand, with design solely based on CFD, there may not really be a need for a standardized system corresponding to the propulsive factors applied today. A probable scenario is that each and every actor may apply its own method to describe the performance of the propulsion system, as long as it serves its purposes.

Using CFD, the complete flow field and forces on components are available for detailed studies which creates possibilities to obtain a higher level of system understanding in relation to what is possible based on model-scale tests. To conduct a CFD study and focusing on the propulsive factors to describe the propulsion system performance, is considered to be a waste of resources, with regards to the effort associated with a representative CFD-model. To summarize the performance in a few overall variables such as the propulsive factors, implies a high risk in losing information, which can complicate the possibilities to understand the important interaction effects. What is rather needed are illustrative evaluation methods for the CFD-results to describe the flow around the propulsion system and the system performance. Such methods are expected to be very useful as a support when understanding and explaining the system performance, both within the internal design process, but also externally with customers and other stakeholders. Well formulated and applied, such methods could also be powerful design tools, supporting the development of propulsion system components more adapted to the system and a higher level of system optimization.

1.3 Purpose and Delimitations

- The objective of this work is to increase the understanding of propulsion system interaction phenomena to support and improve the propulsion system design process.

- To facilitate the understanding of propulsion system interaction phenomena, methods need to be formulated to analyse and describe these for a wider audience.

The main tool used to describe the flow is CFD, applying the RANS equations. The methods used should be possible to apply in an industrial framework, at present or within a near future. The CFD modelling needs to be representative for the system it describes. This work does not include any development of new CFD models, but an important question is:

- How can the the marine propulsion system for the ship and in model-scale be modelled representatively using CFD?

1.4 Composition of the Thesis

This thesis consists of an extended summary and five appended papers. The first objective is achieved through various case studies and is mainly answered in the papers, for a summary of the papers see Section 4. The second and third objectives are summarized in Section 2 and 3, respectively.

The attached papers are linked to the objectives of the thesis in the following manner:

Paper I J. Andersson et al. Energy Balance Analysis of a Propeller in Open Water. *Ocean Engineering* **158** (2018), 162–170

Paper II J. Andersson et al. Energy balance analysis of model-scale vessel with open and ducted propeller configuration. *Ocean Engineering* **167** (2018), 369–379

Describes the foundation and application of the evaluation method based on the control volume approach, i.e. the energy flux analyses. This answers to the second objective, to formulate methods facilitating the understanding of propulsion system interaction phenomena.

Paper III J. Andersson et al. Review and comparison of methods to model ship hull roughness. *Applied Ocean Research* **99** (2020)

Reviews and compares suggested hull roughness models applicable for ship-scale CFD. This is one of the areas that has been in focus to answer the third objective: How can the the marine propulsion system for the ship and in model-scale be modelled representatively using CFD?

Paper IV J. Andersson et al. On the Selection of Optimal Propeller Diameter for a 120-m Cargo Vessel. *Journal of Ship Research* **65**.2 (2021), 153–166

Studies the optimal propeller diameter in-behind in relation to in open water conditions, highly relevant for marine propulsion system design. Hence, the study contributes to the first objective of this work, to increase the understanding of propulsion system interaction phenomena. It does so using the methodology suggested in Paper I and II, i.e. to some extent also answering to the second objective.

Paper V J. Andersson et al. Propeller-Hull Interaction Beyond the Propulsive Factors. *Ocean Engineering (under review)* ()

Studies the in-behind performance of tip unloaded and ice-classed propellers in relation to a more conventional design. The study contributes to the first objective of this work, to increase the understanding of propulsion system interaction phenomena. It does so using surface based evaluation methods, i.e. also answering to the second objective.

2 Describing and Understanding Marine Propulsion System Performance

The main function of the marine propulsion system is to propel the ship forward. How this hydrodynamically is conducted most efficiently could be described applying two separate approaches. The first one states that the ship shall conduct a certain work, i.e. transport a specific amount of cargo at given environmental conditions and a certain speed, with minimal delivered power to the propeller. The delivered power to the propeller is evaluated as the torque multiplied by the rotation rate. The other approach describing the most efficient propulsion system is that the ship conduct a certain work while leaving a minimal trace in the water behind the vessel. A trace is here defined as axial velocities, both positive and negative, transverse velocities, pressure disturbances, increase in turbulence levels, and minor changes in water temperature due to viscous dissipation. These two approaches are central in this thesis since they are the basis for the two main approaches to describe the functioning of a propulsion system. Surface based evaluation methods, as described in Section 2.1, focuses on the forces on the ship and propeller surface, including the torque on the propeller which has a high impact on the delivered power. The control volume approach based on energy fluxes, as described in Section 2.2, instead formulate the delivered power to the propeller in terms of energy fluxes in the water passing the propulsion system.

2.1 Surface Based Evaluation Methods

Surface based evaluation methods stem from the view that the aim of the marine propulsion system is to conduct a certain work at minimized delivered power to the propeller. The delivered power to the propeller is evaluated as the propeller torque multiplied by the rotation rate. Hence it is natural to focus on the torque, or the torque coefficient K_Q. For the propeller operating in a wake the operating conditions for the propeller blade varies significantly around a revolution, as well as radially. Hence, to fully understand the functioning of the propeller blade in the wake, the torque needs to be evaluated at different tangential blade positions, as well as its radial distribution at these positions. Originally the torque stems from the wall shear stress and the pressure distribution around the blade which also is possible to study in detail with the aid of CFD. In Figure 2.1 (from Paper V) K_Q for one blade is plotted around a revolution for three different propellers in both model and ship-scale. These propellers are mounted on a twin-skeg 120 m research vessel. Calculating the deviations from average K_Q, exemplified for propeller C, one blade varies between -44% to +41% and -44% to +33%, in model and ship-scale respectively, due to the varying operating conditions around a revolution. This to some extent illustrates the large variation in operating conditions a blade can experience in a wake, but does not describe the additional radial variations. To study torque variations directly instead of angle of attack, which is a common measure for airfoil or hydrofoil loading, has proven beneficial for propellers operating in a wake. To evaluate angle of attack the hydrofoil relative velocity needs to be defined, which is troublesome in the wake,

the self-propulsion velocity field is disturbed by induced velocities from the propeller and the bare hull velocity field is neither relevant, in addition it is difficult to decide on which axial position to extract it from.

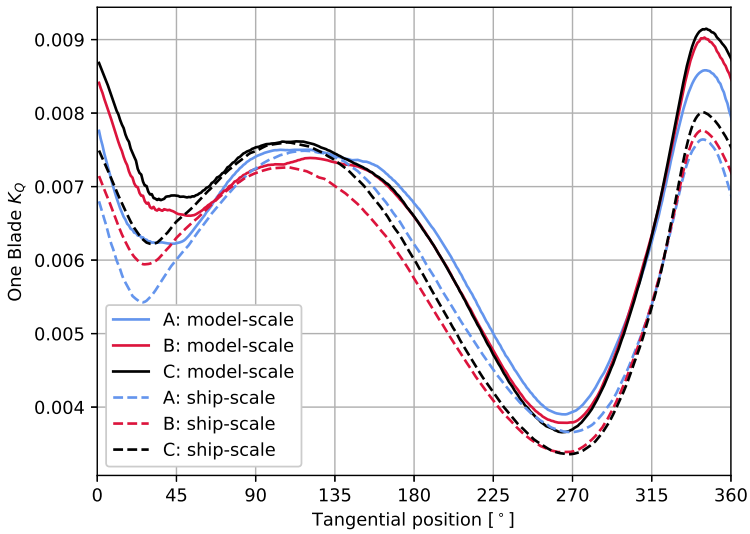

Figure 2.1: *CFD results from Paper V for K_Q variation in the wake.*

In addition to evaluation of torque it is natural to evaluate the thrust on the propeller blades, or the thrust coefficient K_T, and how it varies for the blade around a revolution, as well as radially. Obviously, also the pressure component of the thrust can be studied in more detail through pressure distributions around the blade, as well as the wall shear stress component of the thrust. Anyhow, the propeller thrust is maybe not that useful as such to understand the performance of the propulsion system.

It is not possible to define an efficiency of the propeller operating in-behind conditions, corresponding to the propeller open water efficiency, due to the impossibility to define an advance velocity. Still, to evaluate the performance of the propeller when operating in the wake, some kind of performance measurement is desirable. An alternative measure, suggested in this thesis, is K_T/K_Q. K_T/K_Q is actually the same as thrust over torque divided by radius, which, transformed to the coordinate system commonly applied for airfoils, corresponds to lift to drag ratio (L/D). In other words, K_T/K_Q, or L/D, is not a new measure, but the application of it within ship design to describe and understand propulsion system performance is not yet wide-spread. K_T/K_Q when studied for a blade around a revolution, or focusing on its radial distribution at specific positions, has proven very effective as a measure of the propeller performance in-behind. In Figure 2.2, from Paper V, K_T/K_Q for the three studied propeller blades over a revolution is shown. The differences in K_T/K_Q are naturally influenced by propeller performance in homogeneous

inflow, however two distinct observations could be made based on such a description: The propeller A blade, which is tip-unloaded, has lower K_T/K_Q, i.e. is performing worse, than the other two propellers after the wake peak, between about 0° and 150° and the propeller C blade, which is not ice-classed (the two others are), has lower K_T/K_Q, i.e. is performing worse at minimum load, around 270°. Based on these observations, further analyses of the radial distributions of K_T/K_Q at relevant tangential positions can be conducted. Figure 2.3, also from Paper V, illustrates the radial variation of K_T/K_Q when the blade is at the tangential position 40°, and also the radial variation for a blade operating in open water. Such a figure clearly shows that the performance degradation noted between about 0° and 150° in Figure 2.2 is due to an inferior performance towards the tip for propeller A when operating in the wake. The evaluation of K_T/K_Q is preferably complemented with flow visualizations to illustrate and explain the reasons behind the noted performance differences.

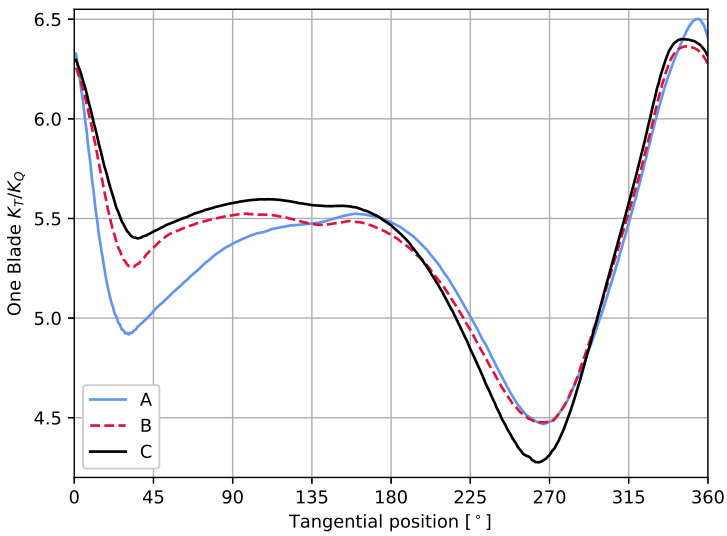

Figure 2.2: *Model-Scale CFD results from Paper V for K_T/K_Q variation in the wake.*

One objective of this thesis is to suggest methods facilitating the understanding of the complete propulsion system performance. The measures suggested so far only focus on the propeller performance in-behind, in other words it needs to be complemented by other data to explain the function of the complete propulsion system. For the surface based evaluation methods it simply corresponds to evaluation of forces, pressure and wall shear stress components, on the other parts of the propulsion system, i.e. the forces balancing the thrust required by the propeller. Relevant components to include are hull, rudder, possible duct or ESD, stags, etc. Also for these forces there is commonly a strong dependency on the blade positions, which can be analyzed further.

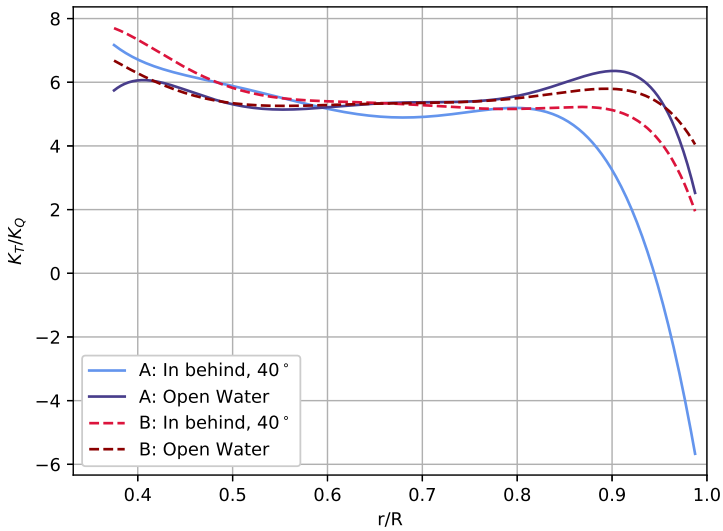

Figure 2.3: *Model-Scale CFD results from Paper V for radial variation of K_T/K_Q at the tangential position 40°. Open water results are at self-propulsion average K_T.*

An important advantage with surface based evaluation methods is that they are simple to evaluate, quantifiable and easily comparable between different users and codes. Nevertheless, they may lack in their ability to describe certain flow phenomena since they focuses on surface responses to the flow and not the flow as such. Therefore surface based evaluation methods are preferably always complemented with visualizations of the flow.

2.2 Evaluation Method Based on a Control Volume Approach

The second approach describing the most efficient marine propulsion system, that the ship is conducting a certain work while leaving a minimal trace in the water behind the vessel, lays the foundation for the control volume approach, based on a balance of energy fluxes, as described in detail in [27].

Different methods are suggested in the literature based on control volume analyses of energy. van Terwisga proposed an energy analysis over a control volume enclosing the entire vessel including propulsion unit [28]. Through the assumption of a uniform control volume inflow, the evaluation of the fluxes were limited to the control volume downstream boundary, but the method was not demonstrated in practice. Schuiling and van Terwisga suggested a methodology for performing an energy analysis based on evaluation of the

energy equation over a control volume, and applied it to a propeller operating in open water as well as in-behind [29, 30]. The viscous losses were obtained through volume integrals of the dissipation terms. Thus, the numerical dissipation, which cannot be evaluated from CFD, was obtained indirectly from the difference between delivered power, obtained from forces acting on the propeller, and the other energy components. Hally [31] has also conducted energy flux analyses, and applied it to a propeller in open water, both with BEM and a full geometrical representation of the propeller. In addition to the complete energy flux balance approaches, it is also possible to study energy fluxes at certain locations to better understand the function of the propulsion system, as for instance applied in Dang et al. [32, 33] and Krasilnikov et al. [34].

Control volume analyses, i.e. application of Reynolds Transport Theorem, is a well established tool, but has traditionally not been applied on CFD simulation results. Reynolds transport theorem states that *the change of any fluid property within the system is the sum of the change within the control volume, plus the outflow from the control volume, minus the inflow to the control volume.* Control volume analyses of energy is actually a power balance, since it is expressed in terms of energy fluxes and can be directly coupled to the delivered power. Control volume analyses of linear momentum, which also traditionally has been applied, can be coupled to thrust and drag. However, studies of these counteracting forces will not provide a relationship to the ship power consumption.

The control volume could be of arbitrary shape, which is of importance to facilitate analyses of various kind of propulsion systems. Figure 2.4 illustrates a possible control volume surrounding skeg, propeller and rudder. The control volume is bounded by both the virtual control surface, as well as the material surfaces, e.g. all or some proportion of the hull, the rudder and the propeller surfaces. To establish an energy flux balance accounting for all propulsive power, the propulsion unit needs to be fully enclosed by the control volume. To increase the understanding of the flow it may be suitable to apply several different control volumes for the same case.

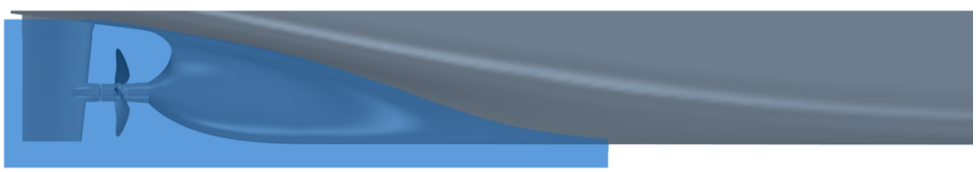

Figure 2.4: *Example of control volume enclosing skeg, propeller and rudder.*

The analysis is based on the energy conservation equation, which reads [35],

$$\Delta E = \dot{Q} - \dot{W}, \tag{2.1}$$

where E represents energy, $\dot{Q}$ denotes the rate at which heat is added to the system and $\dot{W}$ denotes the rate at which work is done by the system. Heat transfer from ship and propulsion unit to surrounding water is preferably neglected, since the associated energy fluxes do not contribute to the hydrodynamic analyses. To handle the fact that the flow around a marine propulsion system is periodical or fully unsteady, energy flux balances

are preferably conducted at several time instances, over a certain period of time. For each time instance it is assumed that the flow is steady, which simplifies the control volume analysis considerably.

Denoting energy per unit mass with e, the energy conservation equation without heat transfer using the Reynolds Transport Theorem for stationary flow yields [35],

$$\Delta E = -\dot{W} = \int_{CS} e\rho(\overrightarrow{V} \cdot \overrightarrow{n})dA, \tag{2.2}$$

where CS denotes the control surface, $\overrightarrow{V}$ the velocity vector, ρ density and $\overrightarrow{n}$ the normal unit vector to the control surface (positive outwards). The work done by the system constitutes work done by pressure and shear stresses on the control surface,

$$\dot{W} = \dot{W}_p + \dot{W}_v = \int_{CS} (p(\overrightarrow{V} \cdot \overrightarrow{n}) - \overrightarrow{\tau} \cdot \overrightarrow{V})dA, \tag{2.3}$$

where p denotes pressure and $\overrightarrow{\tau}$ is the shear stress vector on the elemental surface dA. The pressure and shear stress work acting on the rotating material surfaces of CS constitutes the delivered power (P_D) and can be evaluated as the torque on the propeller multiplied by the rotation rate. Compared to the classical notation, as shown in Eq. 2.1, the delivered power is here defined as power added to the system. Due to no-slip and no flux protruding the hull, no pressure or shear stress work is done by the system on the material surfaces in CS fixed relative to the control volume.

The pressure and shear stress work (Eq. 2.3) also act on the virtual control volume boundaries of CS; these terms are therefore moved to the right hand side of Eq. 2.2 and evaluated together with the energy fluxes. The work done by shear stresses on virtual boundaries of the control volume ($\dot{W}_{v,virtual}$) is at its maximum if the control surfaces are placed tangential to the flow direction and there are significant velocity gradients within the flow. If the flow is approximately normal to the control surface or if they are placed outside the boundary layer, shear stresses are expected to be lower. For all cases studied in Paper I, II and IV, it has been possible to neglect the work done by shear stresses on virtual boundaries of the control volume.

To increase the level of detail in the energy flux balance, the energy per unit mass (e), occurring on the right hand side of Eq. 2.2, is further decomposed. It is proposed to split the term into kinetic energy in axial direction, kinetic energy in transverse directions, internal energy and turbulent kinetic energy:

$$e = \frac{1}{2}V_x^2 + \frac{1}{2}(V_t^2 + V_r^2) + \hat{u} + k, \tag{2.4}$$

where the axial velocity component is denoted by V_x, and tangential and radial velocity components denoted by V_t and V_r, respectively. In a Cartesian coordinate system these components should be replaced with the non-axial velocity components V_y and V_z. The coordinate system is preferably always located so that the axial direction is in line with the vessels sailing direction. This decomposition has been applied within the project leading up to this thesis, but it is of course also possible to separate the contribution from the two transverse velocity components.

Introducing Eq. 2.4 and the above mentioned decomposition of the work rate into Eq 2.2, one gets:

$$P_D = \int_{CS} (\frac{p}{\rho} + \frac{1}{2}V_x^2 + \frac{1}{2}(V_t^2 + V_r^2) + \hat{u} + k)(\overrightarrow{V} \cdot \overrightarrow{n})dA + \dot{W}_{v,virtual}. \tag{2.5}$$

This equation shows that it is possible to express the delivered power as a sum of energy fluxes and rate of pressure work over the surfaces forming the control volume. The internal energy can be obtained through $\hat{u} = c_p T$ (c_p = specific heat capacity, T = temperature), which implies that a temperature field is required for the CFD solution, i.e. the energy equation needs to be solved for.

For an energy analysis to be useful it needs to support the understanding of propulsion system interaction phenomena. This implies that the different terms appearing in Eq. 2.5 need to be fully understood to be able to interpret the results, therefore, the energy flux terms are discussed below.

Rate of Pressure Work and Axial Kinetic Energy Flux

The propulsion unit is converting rotational motion to thrust. A pressure difference is produced between the forward and rear surfaces of the blade and the water is accelerated downstream. This is a continuous energy conversion process where pressure work is converted to axial kinetic energy flux. For the energy flux balance over a propulsion unit, this implies that the distribution between the pressure work and axial kinetic energy flux terms to a large extent will be dependent on the location of the upstream and downstream control surfaces. This energy conversion process is the one which often is explained using an actuator-disc model of a propeller. On the other hand, from the hull point of view the rate of pressure work and axial kinetic energy fluxes originate from flow deceleration and acceleration around the hull.

Firstly, to understand these terms, they are described from a propeller operating in open water point of view: The combined rate of pressure work and axial kinetic energy flux term consists of both useful thrust generation and loss components. This division can be explained through the use of a control volume analysis of both linear momentum and energy, in the manner of Drela [36]. For a propeller in open water, consider a control volume enclosing the propeller, the upstream control surface must be located far upstream so that the inlet conditions can be considered homogeneous, with advance velocity V_A, no tangential flows and undisturbed pressure p_∞, and the lateral surfaces must be streamlines where $p = p_\infty$. For such a control volume the evaluation can be limited to the downstream control surface (*out*) through definition of velocity and pressure perturbations, $\Delta V_x = V_x - V_A$ and $\Delta p = p - p_\infty$. A control volume analysis of linear momentum provides us with the useful thrust,

$$F_x = \int_{out} (\Delta p + (V_A + \Delta V_x)\rho \Delta V_x)dA. \tag{2.6}$$

Through multiplication of all momentum flux balance terms with the advance velocity,

the thrust power is obtained,

$$P_x = F_x V_A = \int_{out} (V_A \Delta p + \rho V_A^2 \Delta V_x + \rho V_A (\Delta V_x)^2) dA. \tag{2.7}$$

Performing a control volume analysis of energy for the same control volume gives the delivered power expressed as a sum of energy fluxes (similar to Eq. 2.5),

$$\begin{aligned} P_D = \int_{out} (\Delta p \Delta V_x + V_A \Delta p + \rho V_A^2 \Delta V_x + \rho V_A (\Delta V_x)^2 + \\ \frac{1}{2}\rho (\Delta V_x)^2 V_x) dA + \int_{out} (\frac{1}{2}(V_t^2 + V_r^2) + \Delta \hat{u} + \Delta k) \rho V_x dA. \end{aligned} \tag{2.8}$$

$\Delta \hat{u}$ and Δk denotes the change over the control volume in internal and turbulent kinetic energy, respectively. Amongst the pressure and axial kinetic energy flux terms in the energy flux balance (Eq. 2.8), the thrust power (Eq. 2.7) can be identified as well as two additional terms, denoted the secondary axial kinetic energy flux,

$$\int_{out} \frac{1}{2}\rho (\Delta V_x)^2 V_x dA, \tag{2.9}$$

and the pressure defect work rate,

$$\int_{out} \Delta p \Delta V_x dA. \tag{2.10}$$

These terms represents the total irreversible outflow losses of pressure work and axial kinetic energy flux through the control volume outlet boundary. They correspond to the total dissipation of pressure work and axial kinetic energy flux to internal energy which eventually occurs downstream due to the mixing out of spatial wake non-uniformity, i.e. the equalizing of pressure and velocity gradients to a homogeneous flow state. Since they arise due to velocity and pressure perturbations and associated velocity gradients in the flow, they are here referred to together as axial non-uniformity losses. However, note that the evaluation of secondary axial kinetic energy flux and pressure defect work rate is not possible for the propeller operating in behind, since neither the lateral control surfaces can be streamlines with $p = p_\infty$, nor the inlet conditions homogeneous.

If instead focusing on a complete ship with propulsion system: The combined sum of rate of pressure work and axial kinetic energy flux for a control volume enclosing the complete ship with propulsion system, should be viewed upon as entirely non useful energy fluxes, i.e. axial non-uniformity losses. This is due to that in the ideal, and only theoretically possible case, the propeller's slipstream, which then is an actuator disk, would completely fill the wake behind the hull such that no axial kinetic energy flux is left behind the ship, as illustrated in Figure 2.5B. This case is obviously not realistic, even for inviscid flow, both since the slipstream cannot completely match the wake, as well as due to that the propeller has a finite number of blades, introducing velocity gradients within the slip-stream.

It is often practical to limit the control volume to the aft part of the ship since a high grid resolution is required for the complete control volume to obtain an accurate balance. However, for such a control volume there needs to be an excess of useful energy flux over the control volume to be able to propel the remaining part of the hull outside the control volume at a constant speed. This implies that a fraction of the sum of rate of pressure work and axial kinetic energy flux terms must be useful thrust power, similar to a propeller operating in open water. The useful amount is not possible to quantify (due to that it is not possible to quantify the advance velocity), a main drawback of the method as a whole. However, a qualitative indication on unnecessary high axial non-uniformity losses are large velocity gradients and zones with high acceleration or deceleration of the flow. Reducing such flow phenomena and designing for an even velocity behind the ship, will reduce the combined sum of rate of pressure work and axial kinetic energy flux.

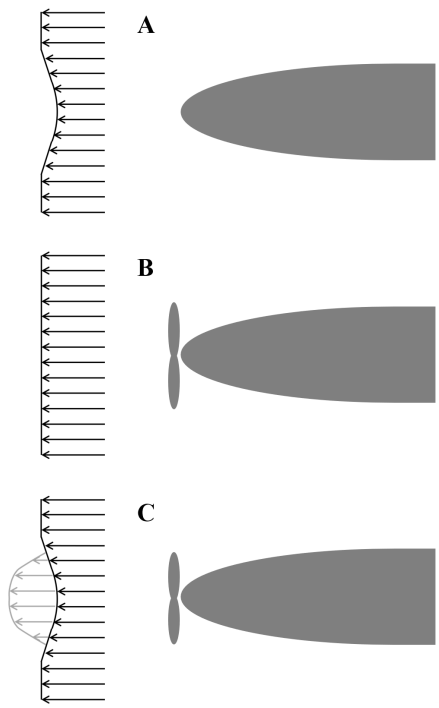

Figure 2.5: *Sketch of the wake of a vessel (A), an ideal actuator disk completely filling that wake (B), illustrating the principle case with zero axial non-uniformity losses, and (C) a general description of a more realistic flow field.*

Transverse Kinetic Energy Flux

Transverse kinetic energy flux is defined as kinetic energy flux in directions other than the vessel sailing direction. Transverse kinetic energy is often associated with radial

and tangential flows induced by the propulsion unit, but can also be due to a propeller slipstream not beeing in line with the sailing direction or bilge vortices caused by the hull curvature. Transverse kinetic energy flux behind the propulsion system should be considered as a loss since the accelerated water in a direction other than the course of the vessel will not contribute to useful thrust.

Internal Energy and Turbulent Kinetic Energy Flux

In a viscous flow, kinetic energy of the mean flow is converted to internal energy, i.e. heat, through two processes: (A) dissipation of turbulent velocity fluctuations and (B) direct viscous dissipation from the mean flow to internal energy. Thus, the internal energy flux is a measure of both these processes, whereas the turbulent kinetic energy flux only accounts for an intermediate stage in (A). The turbulent kinetic energy has to be included only due to the CFD modeling, since turbulence within this project is modeled using an eddy-viscosity model.

All these energy fluxes should be rated as viscous losses, which are highly dependent on boundary layer losses and hence the velocity of the propeller blade relative to surrounding water and wetted surfaces and surrounding water velocities of other components. Also the existence of spatial non-uniformities in the flow, such as circumferential variations associated with the finite number of blades, as well as flow structures like hub and tip vortices, contribute to increased viscous losses when they mix out.

Concluding Remarks on the Control Volume Approach

The main advantage with a control volume approach in relation to a surface based approach is that it really focuses on the flow, not just the response of the flow on surfaces. It was applied successfully in Paper IV, explaining the reasons behind differences in optimal propeller diameter between open water and self-propulsion operation. From the decomposition into different energy fluxes as illustrated in Figure 2.6, the optimum propeller diameter could be explained as a trade-off between blade load/flow acceleration, represented by transverse kinetic energy and axial non-uniformity losses, which increases with decreasing propeller diameter, and viscous losses, increasing with increasing propeller diameter. Comparing this with the case when the propeller operates in isolation, it is possible to deduce that a smaller propeller, with higher transverse kinetic energy losses, gains more from operating together with a rudder, which can straighten up the flow behind the propeller. This implies a larger benefit for propellers of smaller diameter, suffering from higher transverse kinetic energy losses in open water, and motivates a shift towards smaller propeller diameters in behind.

Another beneficial side effect with the energy analysis, when applied as suggested in this thesis, is the solved temperature field, which is commonly not solved for within ship design. The temperature field enables clear visualizations of the viscous losses, which can be addressed for design-improvements. For instance, in Paper II it was shown through contour plots of the internal energy flux downstream the rudder how the end plates of the rudder contributed to increased viscous losses, see Figure 2.7.

Important to note is also that even though surface based evaluation methods are very

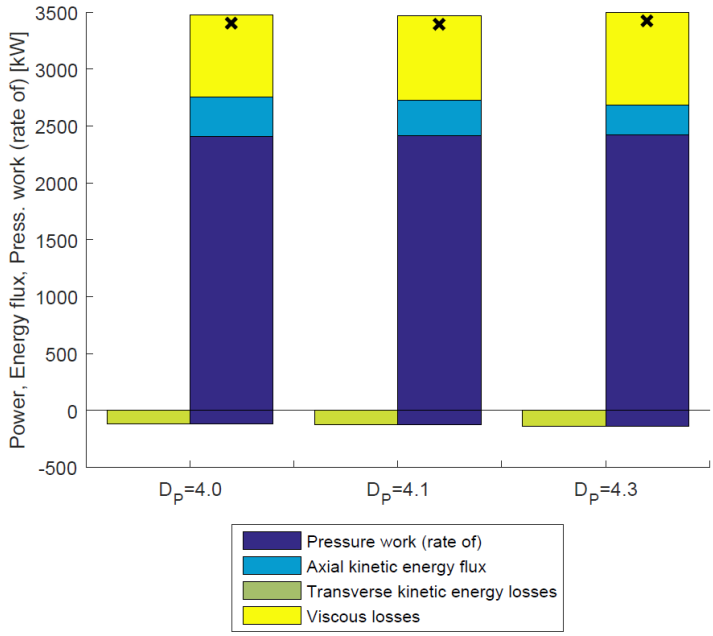

Figure 2.6: *Delivered power decomposed using an energy flux balance for three propellers studied in Paper IV. Power evaluated based on forces on the blade indicated with "x".*

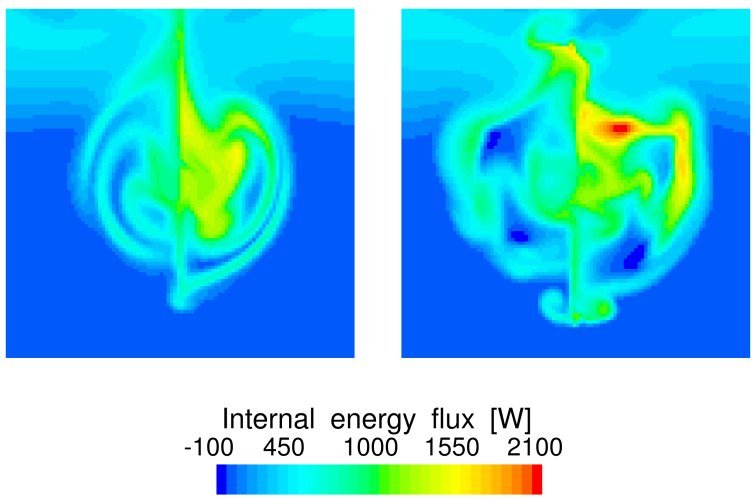

Figure 2.7: *Contour plot of internal energy flux downstream rudder from study conducted in Paper II. Left: open propeller configuration, right: ducted propeller configuration.*

simple to apply, the control volume approach as suggested in this thesis is not that complicated either. It requires a well converged temperature field with high precision of the values, due to the very low temperature increase caused by viscous and turbulent dissipation. It also requires a relatively fine grid within the area of the control volume analyses. The rest of the method, including establishment of control surfaces and definition and evaluation of variables, is just a post-processing step, which preferably is automated.

However, there are clear disadvantages with the control volume approach as well. Firstly, the distribution into the different energy fluxes will be highly dependent on the location of the control volume as well as on the grid refinement. It is therefore necessary to apply identical control volumes and grid refinements when comparing the energy fluxes between different designs, to obtain usable results. A downstream control surface further away from the propulsion system or a coarser grid implies higher viscous losses, to a various extent influenced by increased numerical dissipation. This implies that comparisons of distribution into the different energy fluxes, most probably will be very difficult between different CFD methods. Anyhow, established as a standard procedure within an organization, it will most probably be possible to conduct comparative analyses. A second clear disadvantage, is that for a control volume not surrounding the complete ship, the useful amount of rate of pressure work and axial kinetic energy flux, is not possible to quantify. This implies that only a qualitative indication on the axial non-uniformity losses is possible to obtain.

3 CFD Modelling Aspects of Marine Propulsion Systems

This section aims to summarize some important aspects of CFD modelling of marine propulsion systems considered during the project leading up to this thesis. CFD modelling within ship design has within the research community traditionally been focused on model-scale CFD, a natural consequence of the lack of validation data for ships. An international workshop series on CFD in ship hydrodynamics has been held since 1980 which has contributed considerably to the general knowledge on model-scale CFD [37, 38]. For propeller modelling, model-scale validation workshops on the Potsdam propeller test case (PPTC) has been held at the Symposium on Marine Propulsors in 2011 and 2015 [39, 40]. ITTC also provides summarized guidelines on CFD for ship-design [41] and also more specific on self-propulsion CFD [42].

The focus of the workshops held and the ITTC guidelines has not specifically been on the propulsion system and representative modelling of the interaction effects. An outcome most probably influenced by the lack of detailed data to use for validation and also by the complexity of the flow and higher computational cost associated with detailed propulsion system modelling. As a consequence the general maturity of aspects associated to CFD modelling of the marine propulsion system, in both model-scale and for the ship, is considered lower than for instance for the resistance test or propeller in open water.

The other important area for development is, as earlier mentioned, to increase the general maturity of ship-scale CFD. To facilitate this, more flow data for validation is necessary, which for instance is planned to be obtained through an ongoing industry wide research project [25]. With increasing knowledge on the flow conditions around a ship, more research will most probably be devoted to hull surface modelling and other modelling aspects not yet considered.

An aspect not considered within this project, but of interest for future research is the limitations of RANS and unsteady RANS to capture the flow around marine propulsion systems. In reality the wake is unsteady, in both model-scale and for the ship, leading to instantaneously transient inflow to propeller, rudder and ESD, that might differ considerably compared with the average wake, as discussed in for instance [43]. The influence on airfoil or hydrofoil performance of a turbulent inflow is mainly considered to be characterized by: the reduced frequency, i.e. the inverse to the wave length of the velocity variation; the amplitude of the velocity variation, i.e. the variation of angle of attack; and the mean angle of attack [44, 45]. For wave lengths shorter than the chord length the effective curvature of the foil is modified, in contradiction to wave lengths extending the chord, which can be seen as a sequence of different angle of attacks, i.e. classic foil theories still holds for each instant. The Katzmayr effect, also known as the Knoller-Betz effect, is the ability of a foil to produce thrust in a varying inflow, is valid when the wave length is significantly smaller than the foil chord. Further it is noted that a foil is more sensitive to velocity variations at a high mean angle of attack, were also small velocity variations can trigger separation [46, 45]. The impact of these effects for marine propulsion system performance prediction has not yet been clarified in the

literature, and an important question for future research is whether it is sufficient with a RANS or unsteady RANS model for propeller performance prediction in-behind or to what degree scale-resolving simulations are needed. However, from an engineering perspective, this modelling error is likely to be considered acceptable considering the cost of statistically converged unsteady simulations.

3.1 Modelling Propeller

There is not yet an established best practice for how to model the propeller for various types of propulsion systems or design questions. Common methods includes: virtual propellers using source terms to the momentum equations based on lifting line, lifting surface, or boundary element methods; stationary resolved propellers using multiple reference frames (MRF); and rotating propellers utilizing sliding grid interfaces. The lack of best practice for propeller modelling is exemplified through the CFD benchmark study in Paper H; geometrically resolved propeller models applying sliding mesh interfaces is in average predicting a higher power reduction of a pre-swirl duct installation on KVLCC2 compared to simplified propeller models.

Based on the observations of propeller operating characteristics in-behind when applying a geometrical propeller representation with sliding grid interfaces within this project, two main conclusions are drawn:

- The blade section loading, i.e. the angle of attack, varies significantly both tangentially, as well as radially in the wake. As noted in Figure 2.1 the deviations from average K_Q for one blade in ship-scale was -44% to +33% around a revolution. That variation corresponds to the total K_Q for a blade, individual blade sections will experience even more variations in operating conditions. This example indicates the importance of that the selected propeller model, when placed in an wake, is representative at design as well as off-design conditions, i.e. at both high and low angles of attack. As exemplified in Paper V, differences in performance at off-design conditions may be critical for the in-behind performance.

- The use of a geometrically resolved propeller using MRF is strongly discouraged since it implies that the propeller performance is entirely dependent on its fixed position in the wake. Variations in delivered power of $\pm$ 5 % or more dependent on propeller position can be expected, determined by wake variations, interaction with rudder and number of blades. A stationary geometrically resolved propeller should therefore only be considered for relatively homogeneous inflow conditions without a downstream rudder.

The main reason to avoid the geometrically resolved propeller utilizing sliding grid interfaces is its associated computational cost. The commonly applied strategy to obtain thrust-resistance equilibrium for the ship, also used within this project, is manual variation of the rotation rate, which is time consuming. To reduce the required computational cost, strategies like initialization with MRF, larger time steps in the first phase when applying sliding grids, and possibly starting of with a coarser grid, are recommended.

3.2 Modelling Turbulence

Turbulence modelling is crucial for a correct representation of the boundary layers constituting the inflow to the propulsion unit, as well as for a representative modelling of the components in the propulsion system. It is especially critical in model-scale where the boundary layers are thicker, due to the lower Reynolds number, with higher tendency to separation compared to for the ship. For modelling of the real ship, which actually should be the main focus, turbulence modelling is assumed to be less critical due to the higher Reynolds number and associated thinner boundary layers. Although, since in principal all validation of CFD for ship design so far has been conducted in model-scale, a lot of effort has also been spent on turbulence modelling.

The model-scale hull boundary layers, characterized by an anisotropic turbulence and often rolling up as vortices behind the ship, are challenging to represent accurately for many turbulence models. Within the workshop series on CFD in ship hydrodynamics [37, 38], wake field and resistance have traditionally been evaluated as separate test cases. For wake field it is concluded that linear eddy viscosity models, without ad-hoc rotation correction, in general underestimate the intensity of the bilge vortices. Best performance is seen for various anisotropic turbulence models, such as explicit algebraic stress models (EASM), and Reynolds stress models (RSM) [37, 38]. For resistance prediction the workshop results show that various $k-\varepsilon$ and $k-\omega$ models are more accurate than EASM. Nevertheless, for self-propulsion CFD-predictions it is critical to obtain accurate resistance and wake field predictions using the same turbulence model; to match the towing force applied during test, at the same time as relevant operating conditions for the propeller are established.

For the CFD software applied within this project, STAR-CCM+, a thorough investigation of available turbulence models was conducted in version 12.06 (Paper E). The SST k-ω model [47, 48] with curvature correction [49, 50] and quadratic constitutive relations (QCR) [51] was recommended based on validation using the freely available KVLCC2 and JBC test cases, as well as the ship studied in Papers II and IV. For these hulls the model provides accurate predictions of both model-scale wake field and resistance, and from an engineering perspective it is beneficial that it is stable in terms of convergence and not implies any notable additional computational cost in comparison to more common two-equation turbulence models. Figure 3.1 illustrate the measured and predicted model-scale wake field for KVLCC2 with standard SST k-ω and SST k-ω with curvature correction and QCR. The effect of curvature correction and the QCR model is clearly noted through the differences in bilge vortice prediction on the propeller plane.

The low Reynolds numbers of the flow for model-scale marine propulsion systems, may imply laminar boundary layers and laminar to turbulent transition on the propeller blades and other components. A common turbulence model such as SST k-ω assumes fully turbulent boundary layers and the representativeness of such a model could therefore be questioned. For a propeller operating in open water, with lower upstream turbulence levels, and hence less ability to trigger turbulent boundary layers, this has for a long time been considered as a problem and ITTC recommends to conduct propeller open water tests at minimum Re $= 2 \cdot 10^5$ [52]. Recently, it has been acknowledged that Re $= 2 \cdot 10^5$ most probably not is enough, instead it will be recommended to conduct the

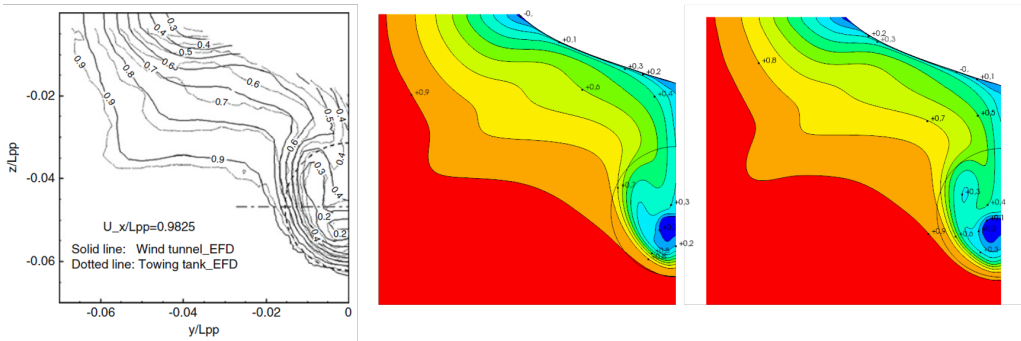

Figure 3.1: *Normalized axial velocity at propeller plane. Experimental data (left), SST $k-\omega$ (middle) and SST $k-\omega$ with curvature correction and QCR (right), both using low-Reynolds-number models ($y^+ < 1$).*

open water tests for each propeller at several different Reynolds numbers to investigate its dependency towards it [53]. The extent of laminar boundary layers and laminar to turbulent transition on the blades in open water can also been verified with paint-streaks in tests, see for instance [17, 18]. Despite that the turbulence levels in the wake are higher, which may trigger turbulence also at lower Reynolds numbers, laminar boundary layers may also be present on the propeller and other components within the marine propulsion system. In a model-scale self-propulsion test the Reynolds numbers are often very low and some parts of the propulsion system may operate in a surrounding flow with less upstream turbulence intensity. However, it is difficult to conduct measurements or other experimental studies of the extent of laminar boundary layers in-behind conditions due to the strong tangential variation in the flow field. Paint-streak tests of the propeller in-behind conditions were conducted in [19], but to which extent these suffered from not being able represent the tangential variation in the flow field is unknown.

Regardless of the lack of detailed data to use for validation, the model-scale self-propulsion CFD analyses in Paper V were conducted applying a transition model, the $\gamma - Re_\theta$ model [54, 55] including the cross-flow term [56] as implemented in STAR-CCM+ ver. 2019.3 [57]. In line with model-scale tests, the boundary layer was triggered to turbulent on a 50 mm wide area on the hull downstream of the bow, representing the sandpaper-stripe. This area was modelled as rough, while the remaining hull was modelled as a smooth wall. The default roughness model implemented in STAR-CCM+ for low-Reynolds number turbulence models [57] was applied together with an equivalent sand grain roughness (k_s) of 300 μm. This roughness height was found appropriate to trigger turbulence, but it was not investigated to which extent it was a correct physical representation of the sandpaper-stripe. The grid on the propeller was adapted to the use of the $\gamma - Re_\theta$ transition model, which implies well resolved boundary layers in all three directions. The grid on the hull was not adapted to the use of a transition model, since the location of laminar-turbulent transition on the hull was controlled by hull roughness. The streamlines on the propeller are shown in Figure 3.2, for a setup with and without transition model respectively. The streamlines on the blades give an indication of the

relative strength of the radial centrifugal force and the tangential wall shear stress. A laminar boundary layer has a lower wall shear stress, allowing the centrifugal force to dominate, resulting in radial streamlines. The other way round, the high shear stress in a turbulent boundary layer causes the streamlines to follow a tangential path. The setup with transition model predicts laminar boundary layers on the propeller blades on both pressure and suction side. The extent of the laminar boundary layers varies around the revolution, where the blade in the top position in the wake peak, seems to have turbulent boundary layers. No detailed characteristics of the propeller boundary layers are available from measurement, but the general trend predicted by CFD seems reasonable.

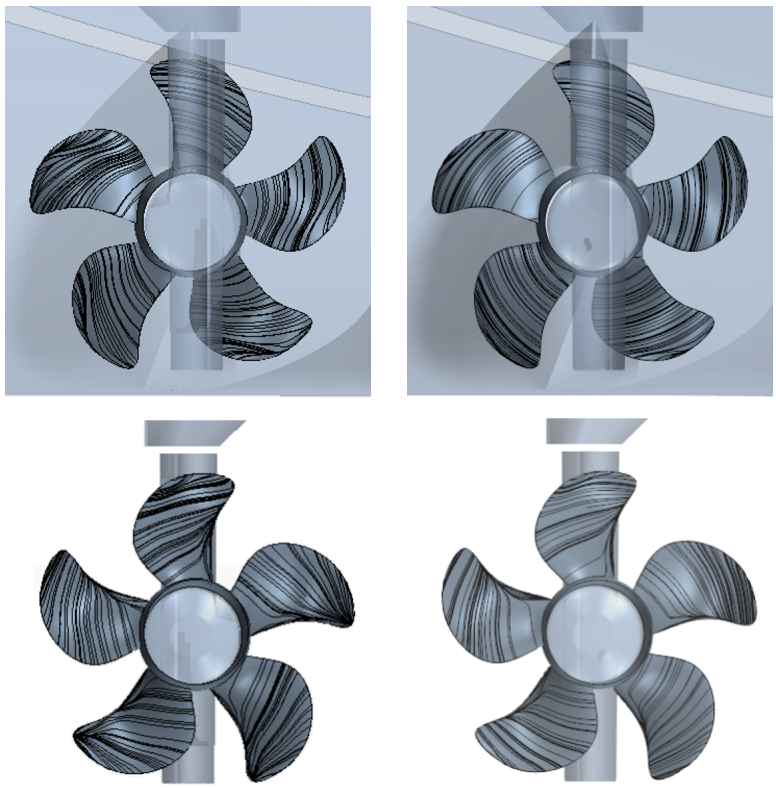

Figure 3.2: *Streamlines on propeller blades for a setup with $\gamma - Re_\theta$ transition model as applied in Paper V (left) and without transition model (right).*

Unfortunately, there is no clear way forward to improve the model-scale CFD modelling of the propulsion system since the extent of the laminar boundary layers on the propeller in general are unknown. The most simple way forward is instead to focus on improving the maturity of CFD modelling in ship-scale, where the problem of unknown extent of laminar boundary layers and laminar to turbulent transition is not an issue.

3.3 Modelling Hull Roughness

In model-scale the surfaces are generally smooth enough to be considered as hydraulically smooth. Modelling of surface roughness has therefore never been an issue for model-scale CFD. For the real ship the surfaces are inevitably rough due to paint roughness and organic growth and also influenced by other imperfections such as welding lines, plate dents, paint defects and plate thickness differences, which all influence the boundary layer development along the ship, critical for the propulsion system inflow.

Recent flow measurements around a ship indicate the necessity to account for hull roughness modelling [23]. However, the scarce amount of data covering both flow and detailed hull surface measurements, implies that this is a modelling aspect with high levels of uncertainty. Considering its large impact on the marine propulsion system performance, as for instance indicated in Paper H, more attention needs to be paid to hull roughness modelling. Important aspects include:

- The standard hull roughness measured on ship hulls is only a measure of the height of the surface roughness, $R_{t,50}$, the maximum peak-to-trough height taken over a 50-mm sample length. It is also known as the Average Hull Roughness (AHR) when combined to one single parameter through averaging several measurements on the hull, according to standardized procedures. For most hull roughness models, as those summarized in Paper III, more detailed surface parameters such as skewness, slope angle, wave length and other measurements of roughness amplitude are often used. For these surface parameters there is no standardized measurement technique and filtering of data established, which actually can influence these values significantly. To facilitate comparison and transfer of knowledge between studies, as well as usage of the results within a wider group, it would be highly beneficial if a standardized procedure to obtain a few specific more detailed surface parameters was agreed upon.

- The review of existing methods to model hull roughness in Paper III shows the use of a variety of roughness functions, both Colebrook-type and inflectional with three distinct flow regimes, as well as a variety of strategies to obtain the roughness length scales. Based on the review, no convergence within the research community towards specific roughness functions or methods to obtain the roughness length scales, was noted. Another recent review [58] claims that at least three roughness parameters are required: one measure on roughness height, one measure on slope, and one on skewness, which seems reasonable. The measures slope and skewness are illustrated in Figure 3.3. More basic and applied research is necessary to increase the maturity and general applicability of roughness models.

- All development of roughness functions and strategies to obtain the roughness length scales, as summarized in Paper III are based on experiments conducted on flat plates without any pressure gradients in the flow, unlike the flow around curved ship hull surfaces. The possible interactions between rough surfaces and flow with pressure gradients need to be studied further.

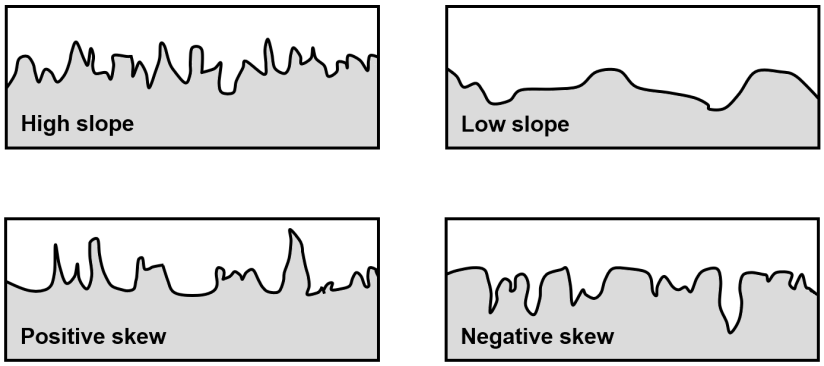

Figure 3.3: *Illustration of the roughness characteristics slope and skewness.*

In addition to the uncertainties associated with the hull roughness modelling there are also uncertainties on how to model and simplify the ship form. Surface imperfections such as weld seams, anodes, larger surface damages, plate dents, and differences between drawing and build geometry, are all details that needs to be taken into consideration. Currently there are very few studies conducted focusing on these issues, but it is suspected that several of these geometrical details are difficult to characterize in a general manner since they are dependent on the location of the imperfections as well as the hull form.

Within this project a mixture of approaches have been applied to model the hull roughness, both based on the review of roughness models in Paper III and reverse engineering type of approaches. The general experience of applying suggested roughness models directly is that they predict a low impact on boundary layers and resistance. If that stems from that the laboratory generated surfaces are not a realistic representation, or if it is due to not considering other surface imperfections, or due to other factors is impossible to deduce without further ship-scale measurements. With a reverse engineering approach, as applied both in Paper IV and V, the equivalent sand grain roughness and associated roughness function are selected such that it produces a specific resistance. The specific additional resistance aimed for, could be taken as the one according to the roughness allowance formula in the ITTC-78 prediction method [15], or an equivalent correlation.

Another important question for the design of marine propulsion systems, which it does not seem to be a consensus in, is which roughness level that should be considered representative for the design point. Is it the one of a clean painted hull or the one of a ship in service? This will most probably appear more important in the future when the knowledge about the ship wake field hopefully is better than today.

4 Summary of Papers

4.1 Paper I

J. Andersson, A. Eslamdoost, A. Capitao-Patrao, M. Hyensjö, and R. E. Bensow. Energy Balance Analysis of a Propeller in Open Water. *Ocean Engineering* **158** (2018), 162–170

Motivation and Division of work

To be able to use energy analyses more than as a conceptual model for an optimal propulsion system a concrete tool is required. This paper describes a control volume analysis of energy applied on a propeller operating in open water. The aim with this simplified case is to investigate influences from control volume size and grid refinement.

All authors participated in stating the aim and scope of the work and contributed with their ideas in how to present the results and structure the paper. I implemented the control volume analysis of energy in post processing scripts, generated the grids, set up the simulations, post processed and analyzed the results and wrote the paper. Alexandre Capito-Patrao contributed with useful knowledge and full derivation of the division between thrust power and non-useful axial kinetic energy and pressure work components.

Results and Conclusion

The paper shows that the delivered power can be expressed with high accuracy using energy fluxes through the control surfaces. The difference between the power evaluated based on energy fluxes and the one based on integrated forces on the propeller surface is less than 1% within the study. The division between the different energy fluxes is found to be grid dependent, so for comparison of different cases, similar level of grid refinement is recommended. The distribution of the energy into the different energy fluxes is shown to be highly dependent on the location of the control volume, and it is therefore important to apply identical control volumes when comparing different cases. More information about the flow is obtained if the downstream control volume surface is located in the vicinity of the object of interest, whereas further away, the kinetic energy terms are to a larger extent converted to internal energy.

Comments

This paper includes the core of the energy flux analysis as suggested in this thesis, although only applied on a propeller in open water. It also clearly illustrates the clear disadvantages with the method, that the division into the different energy components is dependent on both control volume placement, as well as grid refinement. A clear benefit with the method suggested, compared to other published similar methods to study the marine propulsion system, is that it is a straight forward post-processing tool with the only additional requirement of solving the energy equation, and can be employed in any CFD software based on commonly available variables.

4.2 Paper II

J. Andersson, A. Eslamdoost, M. Vikström, and R. E. Bensow. Energy balance analysis of model-scale vessel with open and ducted propeller configuration. *Ocean Engineering* **167** (2018), 369–379

Motivation and Division of work

The energy flux analysis proposed in Paper I was only applied on a propeller in open water, but the purpose of the method is to describe the performance of complete propulsion systems. This paper could be seen as a complement to Paper I, exemplifying how the energy analysis can be applied on a propulsion system or complete ship.

All authors participated in stating the aim and scope of the work and contributed with valuable feedback throughout the project. I generated the grids, set up the simulations, post processed and analyzed the results and wrote the paper.

Results and Conclusion

The paper illustrates the use of different control volumes to analyse the propulsion system. A control volume with relatively tight boundaries around the propeller is considered beneficial for isolated studies of the propeller hydrodynamics but it cannot capture the interaction effects between the hull, propeller and rudder. Naturally, a control volume only has the possibility to describe the flow within it. On the contrary, a control volume can enclose the complete vessel, which requires a higher grid resolution for a larger volume around the vessel. However, the results for such a control volume can be more difficult to grasp and less suitable to use for pedagogical explanations, especially since the axial kinetic energy flux and pressure work consists of positive and negative contributions from wake and propulsor slipstream that cancel each other. The third alternative is to use a control volume enclosing the aft-ship, which is motivated by that the differences between compared propulsion systems to a large extent originate from the aft-ship since the remaining parts are identical, as long as the control volume extends far enough upstream. The drawback of such a control volume, which also holds for a control volume limited to the propeller only, is that an unknown fraction of the pressure work and axial kinetic energy flux terms is associated with the net positive thrust power over the material boundaries enclosing the control volume, i.e. a beneficial and necessary energy flux. In other words, the axial non-uniformity loss cannot be quantified since the thrust power is dependent on an advance velocity which cannot be defined. Despite this, a control volume enclosing the aft-ship is considered to be the most practical configuration for analyzing the system performance in this paper.

The control volume analysis of energy is applied on a model-scale cargo vessel equipped with an open and ducted propeller configuration. The ducted propeller configuration has a much higher required delivered power compared to the open propeller. Through the energy flux analysis it is shown that this, to the largest extent, is due to higher viscous losses, mainly caused by the propeller duct and different rudder configurations. Through solving the energy equation of the flow, which is necessary for the suggested way of

evaluating the internal energy flux, very good visual illustrations of the viscous losses could also be conducted.

Comments

This energy flux analysis does not really provide any general useful design knowledge on propulsion system interaction effects. The contribution of the paper is rather the methodology of applying energy flux balance analyses on complete propulsion systems. If interaction effects would be studied in more detail less design variations should be included between the cases, in this paper three main differences were included: ducted vs. open propeller, hub-cap rudder-bulb system vs. conventional hub-cap, and different rudder designs.

To account for load variations over a blade passage a very rough simplification was carried out in this study. The presented results were obtained through averaging of three different propeller positions. Based on results in later studies, see for instance Paper V, this is not recommended.

As the validation exercise in the paper shows, the CFD results are not fully representative for the tested conditions, especially for the ducted propeller configuration. Aspects such as general turbulence modelling, near wall modelling and laminar-turbulent transition modelling could most probably be reviewed and modified using knowledge gained later within this thesis project.

4.3 Paper III

J. Andersson, D. R. Oliveira, I. Yeginbayeva, M. Leer-Andersen, and R. E. Bensow. Review and comparison of methods to model ship hull roughness. *Applied Ocean Research* **99** (2020)

Motivation and Division of work

Despite the availability of research on the hydrodynamic effects of various hull conditions, it seems like there is little consensus or established best practice within the ship design community on how to model hull roughness for ship-scale CFD. This was clearly noticed in the 2016 Workshop on Ship Scale Hydrodynamic Computer Simulations [21], where only three participants out of 17 included modelling of hull roughness. This was the background to this study with the objective to review and compare different methods to model hull roughness. The study is limited to relatively clean hull surfaces, ranging from high quality newly painted hulls to different extent of poor paint application and/or hull coating damages and light slime layers. More severe biofouling, such as heavy slime layers, weed and calcareous fouling are excluded.

I generated the grids, set up the simulations, post processed and analyzed the results and wrote the paper. Dinis Reis Oliveira, Irma Yeginbayeva and Michael Leer-Andersen especially contributed with their experimental experience and thorough knowledge of hull roughness characterization and modelling. All authors provided me with valuable feedback throughout the project and on the final paper.

Results and Conclusion

From the review it is clear that yet there is no convergence within the research community towards specific roughness functions or methods to obtain the roughness length scales. The comparison of different methods to model hull roughness shows a moderate correlation between additional resistance and AHR, with large scatter due to differences in surface texture, and maybe other properties such as hydrophobicity and surface elasticity, for surfaces with different extent of poor paint application and/or hull coating damages. Amongst the models for high quality, newly painted surfaces no clear correlation between additional resistance and AHR can be noted. Further, for slime layers/biofilms, similar conclusions as drawn in previous studies are made; they are very difficult to characterize and there is a wide variability in their impact on hull resistance.

An important note made in the review is that it can be problematic to compare different hull roughness models and surfaces referred to in the studies since the detailed surface parameters often used are dependent on measurement technique and filtering of data for which there is no standardized method. To facilitate comparison, transfer of knowledge between studies, and usage of the results within a wider group, it would be highly beneficial if the research community agreed upon a standardized procedure to obtain a few specific more detailed surface parameters, such as slope and skewness.

Comments

Despite a lot of research on hull roughness modelling on fabricated surfaces in model-scale there is a critical part remaining to increase the maturity of ship hull roughness modelling: ship-scale flow measurements in combination with hull roughness characterizations. Currently hull roughness modelling is considered as a large uncertainty for both ship resistance and wake prediction, mainly due to often poor knowledge of the detailed hull surface conditions to be modelled, but also due to uncertain validity of selected hull roughness model.

4.4 Paper IV

J. Andersson, R. Gustafsson, A. Eslamdoost, and R. E. Bensow. On the Selection of Optimal Propeller Diameter for a 120-m Cargo Vessel. *Journal of Ship Research* **65**.2 (2021), 153–166

Motivation and Division of work

In the preliminary design of a propulsion unit the selection of propeller diameter is commonly based on open water tests of systematic propeller series. The optimum diameter obtained from the tested propeller series data is not considered to be representative for the operating conditions behind the ship, instead a 2-5% smaller diameter is often selected. The reasons behind this diameter reduction in behind conditions seems to be relatively unknown, or at least not widespread, amongst propeller designers. Therefore this CFD study was initiated to study the reasons behind the conventional reduction of

optimal diameter in behind condition relative to a homogeneous inflow, with focus on understanding the hydrodynamic effects influencing the optimum.

The propulsion systems with varying propeller diameter were designed by Robert Gustafsson, who also initiated the project and contributed with general propeller design knowledge. I generated the grids, set up the simulations, post processed and analyzed the results and wrote the paper. All authors provided me with valuable feedback throughout the project and on the final paper.

Results and Conclusion

For the studied vessel, the CFD results indicate that a 3-4 % smaller diameter is optimal in behind conditions in relation to open water conditions at the same scale factor. Energy flux balances were applied to understand the reasons behind this reduction in optimal propeller diameter. The main reason is assumed to be that smaller propellers to a larger extent benefit from operation with a rudder that can straighten up the propeller slipstream. This can be explained by that a smaller propeller has to be higher loaded over each blade section, i.e. it deflects the flow tangentially to a larger extent, compared to a larger propeller delivering the same power.

Comments

What is not discussed in the paper is that propeller design work most commonly is based on model-scale propeller series data and since the optimal open water propeller diameter in model-scale not is the same as in ship-scale, the commonly used thumb rule seems to be invalid. This study indicates that the difference between model and ship-scale optimal diameter in open water is 7 %, due to reduced viscous losses in ship-scale. In other words, the optimal propeller diameter for the ship is larger, not smaller, than the optimal one from model-scale propeller series data. How much larger is very much dependent on the ship-scale open water performance, which only can be predicted using CFD, and not validated.

Complementing CFD analyses for the ship without a rudder indicates a much larger optimal diameter compared to what is noted with a rudder, supporting the theory presented in the paper, that smaller propellers to a larger extent benefit from operation with a rudder. However, the optimal propeller diameter on the ship without a rudder is still slightly smaller than in open water. This is, at least partly, assumed to be explained by that a smaller propeller to a larger extent can operate in the retarded wake, in relation to a larger propeller.

4.5 Paper V

J. Andersson, R. Gustafsson, R. Johansson, and R. E. Bensow. Propeller-Hull Interaction Beyond the Propulsive Factors. *Ocean Engineering (under review)* ()

Motivation and Division of work

When comparing different propulsion systems based on model-scale tests, differences in the propulsive factors are often not well understood. Two such generic cases of interest to study are tip-unloaded propellers and ice-classed propellers vs. a more conventional design. Through the use of CFD the interaction effects in both model- and ship-scale can be studied and described on a more detailed level. In association to this it is also of interest to evaluate how the traditional propulsive factors can be related to the interaction effects noted, and if this may influence their representability to be used in the scaling procedure.

Robert Gustafsson initiated this project and in addition to contributing with general propeller design knowledge he designed two propellers specifically for this study. I generated the grids, set up the simulations, post processed and analyzed the results and wrote the paper. All authors provided me with valuable feedback throughout the project and on the final paper.

Results and Conclusion

This study utilizes the more straight forward surface based evaluation methods as described in Section 2.1: Evaluation of K_T/K_Q for a blade around a revolution and radial distributions of the same measure, open water performance dependency on Reynolds number and studies of reasons behind thrust/drag differences.

Three main observations on the propulsion system performance in-behind noted are: (1) Tip-unloading is deteriorating propeller performance to a larger extent in-behind conditions since the wake distribution further decreases the load on the blade tips. (2) The blunter leading edge of an ice-classed propeller has a superior performance at low load in relation to a sharper leading edge, it is less sensitive to poor performance at negative angles of attack. This is beneficial in-behind conditions were the load is varying to a large extent. (3) The hull drag is increasing more with a tip-unloaded propeller, hence a more even radial load distribution favour a low thrust deduction factor. Further it is also noted that the model-scale interaction effects to a large extent are influenced by scale-effects between self-propulsion and open water Reynolds numbers. The performance of ice-classed propellers with thicker blade profiles degrades more at low Reynolds numbers.

Even if it can not be determined to what extent the CFD-results correctly predict all aspects of the flow around the self-propelled vessel, the analysis performed indicate that the propulsive factors as derived in model-scale are certainly questionable to use in the scaling procedure. Firstly since they to a large extent are influenced by scale-effects between self-propulsion and open water Reynolds numbers, and secondly since their association to the observed hydrodynamics makes the commonly applied scaling procedure of them doubtful. Scaling of η_O is a separate research topic, but this study indicate that t is not the same in model and ship-scale as commonly assumed and since w_T is not only a measure of the wake velocity, but also influenced by propulsion system performance, a general and representative scaling procedure seems impossible. This study also indicated that η_R is not the same in model and ship-scale, as commonly assumed. η_R seems to be highly dependent on scale-effects between model-scale self-propulsion and open water Reynolds numbers, as well as on the model-scale wake. It shall be kept in

mind that these conclusions are drawn based on one single vessel, and comparison of propellers with and without ice-class and tip-unloaded versus more conventional radial load distributions. For comparison of other propulsion systems, the representativeness of the propulsive factors may not need to be questioned.

Comments

Many may agree on that equal t and η_R in model-scale and for the ship is a very rough assumption, and also that η_O and wake fraction scaling are associated with large uncertainties. Although, there is no simple alternative way forward if ship power predictions are to be based on model-scale tests. For many cases, where similar propulsion systems suffer or gain in the same manner in model-scale, it is not a large problem. It starts to matter when more different propulsion systems are to be compared, such as illustrated in this paper. Hopefully will a more wide-spread knowledge of detailed propulsion system performance increase the awareness of model-scale weaknesses to capture propulsion system performance, and further motivate a shift to ship power predictions based on CFD.

5 Concluding Remarks

This thesis covers several steps taken to improve the general understanding of marine propulsion system performance with the aid of CFD modelling, and also pinpoints a few specific interaction effects to consider when designing marine propulsion systems. As outlined in the introduction, to improve the general understanding of marine propulsion system performance implies a focus on CFD modelling aspects as such, but also on the methods used to describe the flow and its impact on the propulsion system performance.

The first objective was to increase the understanding of propulsion system interaction phenomena to support and improve the propulsion system design process. The studies presented in Papers II, IV and V all include descriptions of such interaction phenomena. The study on optimal propeller diameter for a ship, versus the optimal one based on model-scale propeller series data (Paper IV), is maybe the best example with direct influence on the propulsion system design procedures. The old rule of thumb based on model-scale tests from the 1950s was confirmed by CFD, and supplemented with an explanation: smaller propellers to a larger extent benefit from operation with a rudder that can straighten up the propeller slipstream. Which is due to that a smaller propeller has to be higher loaded over each blade section, i.e. it deflects the flow tangentially to a larger extent, compared to a larger propeller delivering the same power. However, the CFD results also show that the common conclusion most probably only holds within the same scale factor. Hence, the optimal propeller diameter for the ship is most probably larger than what is indicated by model-scale propeller series data for open water conditions. Paper V highlights the importance of high propeller efficiency also in off-design conditions, for this study specifically at low load, since the blade partially will experience these operating conditions when operating in the wake. Further, Paper II illustrates how a small deterioration in performance of a component, for instance the rudder, is amplified in a system: If a modified rudder design causes a slight resistance increment the propeller needs to produce higher thrust to obtain thrust-resistance equilibrium. This implies a marginally lower propeller efficiency, as well as a slightly larger suction on the hull impairing the pressure recovery. The rudder itself will also be exposed to a slightly higher slip-stream velocity, further increasing the resistance of the rudder. This last observation really motivates the importance of optimizing all components when designing the propulsion system. Naturally there are many other generic interaction effects of importance for several systems deserving to be studied and explained. Further studies based on CFD, preferably taking into account existing knowledge based on model-scale test and design experience, are strongly encouraged.

The second objective was to facilitate the understanding of propulsion system interaction phenomena through formulation of methods that could be used to describe these for a wider audience. As clear from Section 2 a variety of methods have been proposed and applied within the project. One is not deemed to be better than another, they rather suit various purposes. The surface based evaluation methods suggested within this project are simple to apply and, to a large extent, possible to compare universally between different users and codes. Since the surface forces can be decomposed on a detailed level using CFD they may in some cases be very useful to understand the system performance. However, they may lack in their ability to describe certain flow phenomena since they

focus on surface responses to the flow and not the flow as such, therefore they are preferably always complemented with visualizations of the flow. The key difference, and main benefit, of energy flux analyses in relation to surface based evaluation methods is really the focus on the flow field instead of on forces on the material surfaces to understand the functioning of the system. However, the decomposition into different energy fluxes will be dependent on CFD-setup, grid resolution and control volume and it may also be difficult to separate the useful energy fluxes from the non-useful ones. Therefore energy flux analyses are less suitable for universal comparisons of interaction effects and they are neither deemed suitable for automatic optimization algorithms. Despite that energy flux analyses really focus on the flow field, they are preferably also always complemented with visualizations of the flow to be able to pinpoint possible improvements of the systems or certain flow phenomena. Further studies on various methods to explain the propulsion system performance, including development of the ones described in this project, are warmly welcomed. Since their aim is to support development of more efficient propulsion systems, it needs to be conducted by or in close cooperation with the industry, to ensure that they serves their purpose.

The third objective regards further knowledge and suitable application of CFD models to characterize the propulsion system performance in a representative manner. The progress made within this area is described in Section 3, but an important part is missing. The aim of the propulsion systems is to propel the ship efficiently, and to assess that representative CFD models in ship-scale are required. Within this project ship-scale flow data has not been available, which really is necessary to further develop and increase the maturity of ship-scale CFD. Ship-scale measurements are especially important to sort out a lot of open questions associated to the wake field and hull roughness modelling. To conduct model-scale CFD may still be relevant to explain performance differences noted in test, but with too much focus on it comes also the risk to get stuck in complicated turbulence and transition modelling, instead of spending the resources on the system of highest importance, the real ship.

Both nationally and internationally several research and commercial studies are ongoing on the design of marine propulsion systems. My belief is that the most important aspects to focus on in the future to achieve more efficient propulsion systems are both increased knowledge on the ship-scale performance at various operating conditions as well as an increased general understanding of propulsion system interaction effects. Increased knowledge on ship-scale operating conditions includes both flow measurements including detailed hull surface characterization, as well as influences from operation in waves, interaction with surface, and performance at various speeds. More studies on propulsion system interaction effect will hopefully increase the awareness of model-scale weaknesses to capture propulsion system performance, and further motivate a shift to ship power predictions based on CFD. The energy flux analyses clearly illustrates that a system being optimal in model-scale most probably is not optimal in ship-scale since the trade off between viscous losses and axial non-uniformity/transverse kinetic energy losses will be different.

Currently the ship market structure in many cases does not favor a high level of propulsion system optimization. When different corporations deliver the separate components within the propulsion system and another one designing the hull, the complete propul-

sion system cannot be optimized as a unit if there is unwillingness to share geometrical details with possible competitors. Another factor influencing the level of optimization is the short development times associated with the tailor-made design for each vessel. This is commonly overcome through larger series of sister vessels or standardized designs. These negative influences from the market structure clearly shows that marine propulsion system optimization is not only a technical challenge, but also an managerial one, and the motivating factor required by both are increased fuel prices and/or legislation.

References

[1] J. Andersson, A. Eslamdoost, A. Capitao-Patrao, M. Hyensjö, and R. E. Bensow. Energy Balance Analysis of a Propeller in Open Water. *Ocean Engineering* **158** (2018), 162–170.

[2] J. Andersson, A. Eslamdoost, M. Vikström, and R. E. Bensow. Energy balance analysis of model-scale vessel with open and ducted propeller configuration. *Ocean Engineering* **167** (2018), 369–379.

[3] J. Andersson, D. R. Oliveira, I. Yeginbayeva, M. Leer-Andersen, and R. E. Bensow. Review and comparison of methods to model ship hull roughness. *Applied Ocean Research* **99** (2020).

[4] J. Andersson, R. Gustafsson, A. Eslamdoost, and R. E. Bensow. On the Selection of Optimal Propeller Diameter for a 120-m Cargo Vessel. *Journal of Ship Research* **65**.2 (2021), 153–166.

[5] J. Andersson, R. Gustafsson, R. Johansson, and R. E. Bensow. Propeller-Hull Interaction Beyond the Propulsive Factors. *Ocean Engineering (under review)* ().

[6] J. Andersson, M. Hyensjö, A. Eslamdoost, and R. E. Bensow. "CFD Simulations of the Japan Bulk Carrier Test Case". *Proceedings of the 18th Numerical Towing Tank Symposium*. Cortona, Italy, 2015.

[7] J. Andersson. *CFR 137: Analyzing Ship Propulsion Interaction Effects - Review of Published Methods*. Internal RR-report, 2015.

[8] A. Eslamdoost, J. Andersson, R. Bensow, R. Gustafsson, and M. Hyensjö. "Analysis of Propeller-Hull Interaction Phenomena on a Self-Propelled Axisymmetric Body". *Proceedings of the Fifth International Symposium on Marine Propulsors*. Espoo, Finland, 2017.

[9] J. Andersson, R. Gustafsson, A. Eslamdoost, and R. E. Bensow. "On the Selection of Optimal Propeller Diameter for a 120m Cargo Vessel". *SNAME Propeller-Shafting 2018 Symposium*. Norfolk, Virginia, USA, 2018.

[10] J. Andersson. *CFR 168: Turbulence Model Evaluation Using the KVLCC2 Test Case*. Internal RR-report, 2019.

[11] A. Eslamdoost, J. Andersson, R. Bensow, and M. Vikström. "Application of Control Volume Energy Balance for Analysing Propeller-Hull Interaction in Presence of Free-surface". *Proceedings of the Sixth International Symposium on Marine Propulsors*. Rome, Italy, 2019.

[12] J. Andersson, R. Gustafsson, R. Johansson, and R. E. Bensow. "In-Behind Performance of an Ice Classed Propeller in Model and Ship-Scale". *Proceedings of IX International Conference on Computational Methods in Marine Engineering, MARINE 2021*. Online, 2021.

[13] J. Andersson, A. Abolfazl Shiri, R. E. Bensow, J. Yixing, W. Chengsheng, Q. Gengyao, G. Deng, P. Queutey, Y. Xing-Kaeding, P. Horn, T. Lucke, H. Kobayashi, K. Ohashi, N. Sakamoto, F. Yang, Y. Gao, B. Windén, M. G. Meyerson, K. J. Maki, S. Turnock, D. Hudson, J. Banks, M. Terziev, T. Tezdogan, F. Vesting, T. Hino, and S. Werner. Ship-Scale CFD Benchmark Study of a Pre-Swirl Duct on KVLCC2. *Applied Ocean Research (under review)* ().

[14] IMO. *Reducing greenhouse gas emissions from ships*. 2019. URL: https://www.imo.org/en/MediaCentre/HotTopics/Pages/Reducing-greenhouse-gas-emissions-from-ships.aspx.

[15] ITTC. *1978 ITTC Performance Prediction Method. Recommended Procedure 7.5 - 02 - 03 - 01.4 Rev 04*. 2017.

[16] ITTC. *Resistance Test. Recommended Procedure 7.5-02-02-01*. 2011.

[17] A. Bhattacharyya, J. C. Neitzel, S. Steen, M. Abdel-Maksoud, and V. Krasilnikov. "Influence of Flow Transition on Open and Ducted Propeller Characteristics". *Proceedings of the Fourth International Symposium on Marine Propulsors*. Austin, Texas, USA, 2015.

[18] D.-Q. Li, P. Lindell, and S. Werner. "Transitional flow on model scale propellers and their likely influence on performance prediction". *Proceedings of the Sixth International Symposium on Marine Propulsors*. Rome, Italy, 2019.

[19] N. Hasuike, M. Okazaki, A. Okazaki, and K. Fujiyama. "Scale effects of marine propellers in POT and self propulsion test conditions". *Proceedings of the Fifth International Symposium on Marine Propulsors*. Espoo, Finland, 2017.

[20] G. Dyne. "The Principles Of Propulsion Optimization". *Trans. RINA 137*. London, United Kingdom, 1995.

[21] D. Ponkratov. *Proceedings of 2016 Workshop on Ship Scale Hydrodynamic Computer Simulation*. Southampton, 2017.

[22] Y. Inukai, Y. Sudo, H. Osaki, T. Yanagida, M. Mushiake, and S. Kawanami. "Extensive Full-Scale Measurement on Propeller Performance of 14000 TEU Container Ship". *3rd Hull Performance & Insight Conference*. Redworth, UK, 2018, pp. 27–35.

[23] N. Sakamoto, H. Kobayashi, K. Ohashi, Y. Kawanami, B. Windén, and H. Kamiirisa. An overset RaNS prediction and validation of full scale stern wake for 1,600TEU container ship and 63,000 DWT bulk carrier with an energy saving device. *Applied Ocean Research* **105** (2020).

[24] T. Wakabayashi, Y. Inukai, T. Yonezawa, N. Igarashi, M. Mushiake, and S. Kawanami. "Full-Scale Measurement of a Flow Field at Stern using Multi-Layered Doppler Sonar (MLDS)". *4th Hull Performance & Insight Conference*. Gubbio, Italy, 2019, pp. 172–178.

[25] JORES. *Development of an industry recognised benchmark for Ship Energy Efficiency Solutions*. 2019. URL: https://jores.net/.

[26] MARINE. *Announcement - Wageningen 2022*. 2021. URL: https://www.w2022.nl/announcement/.

[27] J. Andersson. *Using Enegy Fluxes to Analyze the Hydrodynamic Performance of Marine Propulsion Systems*. Licentiate Thesis, Chalmers, 2018.

[28] T. van Terwisga. "On the working principles of Energy Saving Devices". *Proceedings of the Third International Symposium on Marine Propulsors*. Launceston, Tasmania, Austrailia, 2013.

[29] B. Schuiling and T. van Terwisga. "Energy Analysis of a Propeller in Open Water Using a RANS Method". *24th International HISWA Symposium on Yacht Design and Yacht Construction*. Amsterdam, The Netherlands, 2016.

[30] B. Schuiling and T. van Terwisga. "Energy Loss Analysis for a Propeller operating behind a Ship". *32nd Symposium on Naval Hydrodynamics*. Hamburg, Germany, 2018.

[31] D. Hally. "Energy Balance Analysis using RANS-BEM coupling". *Proceedings of the Sixth International Symposium on Marine Propulsors*. Rome, Italy, 2019.

[32] J. Dang, G. Dong, and H. Chen. "An Exploratory Study on the Working Principles of Energy Saving Devices (ESDS) - PIV, CFD Investigations and ESD Design Guidelines". *Proceedings of the ASME 2012 31st International Conferecen on Ocean, Offshore and Arctic Engineering OMAE2012*. Rio de Janeiro, Brazil, 2012.

[33] J. Dang, C. Hao, L. Rueda, and H. Willemsen. "Integrated Design of Asymmetric Aftbody and Propeller for an Aframax Tanker to Maximize Energy Efficiency". *Proceedings of the Fourth International Symposium on Marine Propulsors*. Austin, Texas, USA, 2015.

[34] V. Krasilnikov, K. Koushan, M. Nataletti, L. Sileo, and S. Spence. "Design and Numerical and Experimental Investigation of Pre-Swirl Stators PSS". *Proceedings of the Sixth International Symposium on Marine Propulsors*. Rome, Italy, 2019.

[35] F. M. White. *Fluid Mechanics*. Sixth Ed. New York: McGraw-Hill, 2008.

[36] M. Drela. Power Balance in Aerodynamic Flows. *AIAA* **47**.7 (2009), 1761–1771.

[37] L. Larsson, F. Stern, and M Visonneau. *Numerical Ship Hydrodynamics – An assessment of the Gothenburg 2010 Workshop*. Dordrecht: Springer, 2014.

[38] T. Hino, F. Stern, L. Larsson, M. Visonneau, N. Hirata, and J. Kim. *Numerical Ship Hydrodynamics: An Assessment of the Tokyo 2015 Workshop*. Cham: Springer, 2020.

[39] M. Hoekstra, T. van Terwisga, H.-J. Heinke, L. Lübke, M. Abdel-Maksoud, and M. Druckenbrod. *Proceedings of the Workshop on Cavitation and Propeller Performance SMP 2011*. Hamburg, Germany, 2011.

[40] S. Kinnas, M. Abdel-Maksoud, U. Barkmann, L. Lübke, and Y. Tian. *Proceedings of the Second Workshop on Cavitation and Propeller Performance SMP 2015*. Austin, Texas, USA, 2015.

[41] ITTC. *Practical Guidelines for Ship CFD Applications ITTC 7.5-03-02-03*. 2014.

[42] ITTC. *Practical Guidelines for Ship Self-Propulsion CFD 7.5-03-03-01*. 2014.

[43] R. E. Bensow. "Large eddy simulation of a cavitating propeller operating in behind conditions with and without pre-swirl stators". *Proceedings of the 4th International Symposium on Marine Propulsors - SMP'15*. Austin, Texas, USA, 2015.

[44] E. G. Paterson and F. Stern. Computation of Unsteady Viscous Marine-Propulsor Blade Flows — Part 2 : Parametric Study. *Journal of Fluids Engineering* **121**.March 1999 (1999), 139–147.

[45] E. Löfgren. *CFR-120: Accuracy of transient versus steady state forces on a rudder operating in a propeller slipstream*. Internal RR-report, 2015.

[46] L. Gilling, N. N. Sørensen, and L. Davidson. "Detached Eddy Simulations of an Airfoil in Turbulent Inflow". *Proceedings of the 47th AIAA Aerospace Sciences Meeting*. Orlando, Florida, USA, 2009.

[47] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal* **32**.8 (1994), 1598–1605.

[48] F. R. Menter, M. Kuntz, and R. Langtry. "Ten Years of Industrial Experience with the SST Turbulence Model". *Turbulence, Heat and Mass Transfer 4*. Ed. by K Hanjalic, Y Nagano, and M Tummers. Begell House, Inc., 2003.

[49] Siemens PLM Software. *STAR-CCM+ Documentation version 12.06*. 2017.

[50] S. K. Arolla and P. A. Durbin. Modeling rotation and curvature effects within scalar eddy viscosity model framework. *International Journal of Heat and Fluid Flow* **39** (2013), 78–89.

[51] P. R. Spalart. Strategies for turbulence modelling and simulations. *International Journal of Heat and Fluid Flow* **21**.3 (2000), 252–263.

[52] ITTC. *Open Water Test - 7.5-02-03-02.1*. 2014.

[53] R. Pattenden, N. Lemb Larsen, J. L. Dantas, B. Metcalf, W. Wang, Y. Inukai, T. Katsui, S. Cheon Go, H. Ahn, P Queutey, D. B. Danisman, Y. K. Demirel, and A. Yakovlev. *The Resistance and Propulsion Committee: Final Report and Recommendations to the 29th ITTC*. 2021.

[54] R. B. Langtry, F. R. Menter, S. R. Likki, Y. B. Suzen, P. G. Huang, and S. Volker. A Correlation-Based Transition Model Using Local Variables Part II: Test Cases and Industrial Applications. *Journal of Turbomachinery* **128**.3 (2006), 423.

[55] F. R. Menter, R. B. Langtry, S. R. Likki, Y. B. Suzen, P. G. Huang, and S Völker. A Correlation-Based Transition Model Using Local Variables Part I: Model Formulation. *Journal of Turbomachinery* **128**.3 (2006), 413–422.

[56] R. B. Langtry, K. Sengupta, D. T. Yeh, and A. J. Dorgan. "Extending the γ-Reθt Local Correlation based Transition Model for Crossflow Effects". *45th AIAA Fluid Dynamics Conference*. Dallas, USA, 2015, pp. 1–12.

[57] Siemens PLM Software. *STAR-CCM+ Documentation version 2019.3*. 2019.

[58] D Chung, N Hutchins, M. P. Schultz, and K. A. Flack. Predicting the Drag of Rough Surfaces. *Annual Review of Fluid Mechanics* **53** (2021), 439–471.