THESIS FOR THE DEGREE OF LICENTIATE OF ENGINEERING IN THERMO AND FLUID DYNAMICS

Using Energy Fluxes to Analyze the Hydrodynamic Performance of Marine Propulsion Systems

JENNIE ANDERSSON

Department of Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2018

Using Energy Fluxes to Analyze the Hydrodynamic Performance of Marine Propulsion Systems
JENNIE ANDERSSON

Thesis for the degree of Licentiate of Engineering 2018:23
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 2018

Using Energy Fluxes to Analyze the Hydrodynamic Performance of Marine Propulsion Systems

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

Abstract

The strive towards more fuel efficient ships is a continuously ongoing process, motivated by both economic and regulatory reasons. An important aspects to consider for the final fuel consumption is the propulsion system performance in relevant operating conditions. The propulsion system performance is most commonly described using a well-established terminology, including thrust deduction, wake fraction, and propulsive efficiency, a decomposition with its primary origin in the experimental procedures used to establish ship scale performance. Since this decomposition does not really provide us with any details about the flow, it can imply limitations in design and optimization of the propulsion system, as the interaction thus may not be correctly represented and fully understood.

Numerical methods, such as Computational Fluid Dynamics (CFD) based on the Reynolds-Averaged Navier-Stokes (RANS) equations, can be used to extract detailed data of the flow around marine propulsion systems. It is proposed in this thesis to conduct control volume analysis of energy based on CFD results to describe the performance of the propulsion system. 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. Through a decomposition of the energy fluxes over the control volume surface the system performance can be described in terms of kinetic energy in axial direction, rate of pressure work, kinetic energy in transverse directions, internal energy, and turbulent kinetic energy, the two last representing the viscous losses.

In general there are no restrictions of how to construct the control volume, it rather depends on the analysis objectives, for which it needs to enclose the entire flow domain of interest. However, it is shown that the downstream surface preferably is located in the vicinity of the studied object, to obtain more details of the flow before it has dissipated into internal energy. Further, from conducted studies it is clear that the control volume for flexibility preferably is constructed in the post-processing phase. It is evident that the possibility to characterize the flow is entirely dependent on the underlying CFD solution, which needs a sufficiently refined grid and suitable models to accurately capture the flow field around the propulsion system. Important aspects to consider for the propulsion system modelling discussed within the thesis are: representation of hull boundary layers, interaction with the free surface, the possible influence from laminar boundary layers on the propeller (model scale), and surface roughness of both hull and propeller (ship scale). The control volume analysis of energy has within this project successfully been applied to describe the performance differences for a vessel operating with an open and ducted propeller respectively, and for describing the reasons behind the established optimal propeller diameter reduction in behind conditions relative a homogeneous inflow.

Keywords: Propulsor-Hull interaction, Control volume analysis, Energy balance analysis, CFD, RANS

List of Publications

This thesis consists of an extended summary and the following appended papers:

Paper I 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 II 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 III 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 IV 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

Acknowledgements

This work had not been possible without all the great people that have surrounded me and provided advice, support, 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 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, Kåre Nerland, Rikard Johansson, Göran Grunditz, Johan Lundberg, Geir-Åge Øye and Kai-Egil Olsen for engaging in my project. I would also like to acknowledge my colleagues at the department of Mechanics and Maritime sciences, both at campus Lindholmen and campus Johanneberg, for providing an inspiring working environment. My co-supervisor Arash Eslamdoost, for careful proofreading throughout, Alexandre Capitao-Patrao for fruitful discussions concerning energy flux balance analyses and Daniel Lindblad for his never ending interest in my project. Finally, I would like to thank my very patient husband Marcus for all support and tolerance with my ups and downs, as well as our son Isak for his inspiring and never-ending positive view of life.

This research was supported by the Swedish Energy Agency (grant number 38849-1) and Rolls-Royce Marine through the University Technology Centre in Computational Hydrodynamics hosted by the Department of Mechanics and Maritime Sciences at Chalmers. This 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
e	Energy per unit mass
D_P	Propeller diameter
k	Turbulent kinetic energy
M	Moment, torque
n	Rotation rate
$\overrightarrow{n}$	Normal unit vector
P_D	Delivered power
p	Pressure
$\dot{Q}$	Heat transfer rate
T	Temperature
$\hat{u}$	Internal energy
$\overrightarrow{V}$	Velocity vector
V_A	Advance 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

Greek symbols

ρ	Density
$\overrightarrow{\tau}$	Shear stress vector

Abbreviations

CFD	Computational Fluid Dynamics
CS	Control volume surface
CV	Control volume
EASM	Explicit algebraic stress model
ITTC	International Towing Tank Conference
JBC	Japan bulk carrier
MRF	Multiple reference frames
RANS	Reynolds Averaged Navier-Stokes
RSM	Reynolds stress model
VOF	Volume of fluid

Contents

1 Introduction

The strive towards more fuel efficient ships is a continuously ongoing process, motivated by both economic and regulatory reasons. In the hydrodynamic design, important aspects to consider for the final fuel consumption are both ship resistance and the marine propulsion system performance, in relevant operating conditions.

A marine propulsion unit is often operating in the wake or boundary layer of the ship it is mounted on. This implies that the propulsion unit affects the flow around the ship, and the inflow to the propulsion unit 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, operating together with the ship, is what will be referred to as marine propulsion system.

Most often, each component in the propulsion system performs very well individually or in a standard system, but the knowledge about how the system is performing as a whole, including potential improvements, are generally not as widespread. This is to some extent a consequence of the market structure, with different companies often delivering the separate components within the propulsion system. However, in the future the strive towards more fuel efficient ships will hopefully help to solve the market structure restrictions, and support a more complete system perspective, earlier in the design phase, to better optimize the tightly coupled components. Often, separate designs of the optimal propeller in a given wake of a hull, and the hull shape with the lowest resistance, do not constitute the most optimal system together. It is therefore believed that there is a large potential of reduced fuel consumption through such system optimization, in relevant operating conditions, early in the design phase.

The flow around the marine propulsion system is complex, the propeller is often operating in a turbulent wake, each propeller blade may experience substantially different operating conditions during a revolution, and the operation is influenced by and influences the water surface. This complexity of the flow implies that the design of marine propulsion systems historically mainly has been supported by model scale testing in basins for verification of system performance and ship scale predictions. The validity of model basin test as being representative for ship scale conditions may be questioned for some propulsion systems, since Reynolds number similarity cannot be kept between model and ship scale, for practical reasons. However, numerical tools have historically not been an alternative, due to the computational resources required to capture the complex flow pattern around the propulsion unit in the wake of a vessel. During the last decades it has become more and more feasible to complement the model basin tests with various numerical methods, which therefore have increased in both maturity and popularity as design tools for marine propulsion systems.

For model scale testing, the amount of information about the marine propulsion system performance is limited to what is possible to measure. Included in the list of measurements are most often velocity field measurements, at least for the bare hull, and overall variables, such as delivered thrust and torque in open water and self-propulsion

conditions, and bare hull resistance. The interaction effects between hull and propulsion system are most commonly described based on these measured overall variables, i.e. they can describe the performance of the propeller in self-propulsion in relation to open water conditions, and the resistance of the hull with propeller in relation to without. The terminology evaluated based on the measured overall variables includes: thrust deduction, wake fraction, hull efficiency, relative rotative efficiency and propulsive efficiency. Since this decomposition into various interaction effects has its primary origin in the experimental procedures used to establish ship scale performance rather than from principles of hydrodynamics, it does not really provide any details about the flow. This can imply limitations in design and optimization of hull and propulsion system, as the interaction thus may not be correctly represented and fully understood.

Numerical methods, such as Computational Fluid Dynamics (CFD) based on the Reynolds-Averaged Navier-Stokes (RANS) equations, can be used to extract detailed data of the flow around marine propulsion systems. This opens up new possibilities to characterize the interaction effects based on principles of hydrodynamics, hopefully describing the functioning of the systems in a more pedagogical manner. Naturally, such analyses based on CFD require well resolved and reliable numerical results, representative for the real flow conditions, in both model and ship scale.

1.1 Purpose

The vision guiding the work leading up to this thesis has been to create tools enabling design of more energy efficient ships. Such a tool require both an effective CFD methodology for ship propulsion systems, as well as a methodology for analyzing the numerical results. The purpose of this thesis is to describe a possible method to analyze and characterize a marine propulsion system, using control volume analysis of energy, as well as to summarize some important aspects to consider when simulating the flow around marine propulsion systems.

2 Obtaining and Interpreting Energy Fluxes for a Marine Propulsion Systems

Different methods for analyzing marine propulsion systems and interaction effects between the different components based on CFD or other calculated results have earlier been proposed in the literature. Dyne [5] suggested a propulsive efficiency based on wake losses and gains. The method was derived based on potential flow assumptions, which implies that it is not applicable for analyzing viscous flow simulation results. However, it is an appealing idea and easily understandable concept to separate the flow features in losses and gains. Dang et al. [6, 7] evaluated the dimensionless kinetic energy in the wake for comparison of different propulsion systems. This methodology focuses on axial and transverse kinetic energy, i.e. not accounting for all energy transferred from the propeller to the water. A more comprehensive methodology was proposed by van Terwisga [8] based on an energy balance over a control volume enclosing the entire vessel including propulsion unit. Through the assumption of a uniform control volume inflow, the evaluation of the fluxes were limited to the control volume downstream boundary. However, the method was not demonstrated. Schuiling and van Terwisga [9, 10] suggested a methodology for performing an energy analysis based on evaluation of the energy equation over a control volume, and applied it on a propeller operating in open water as well as in behind. 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.

Interaction effects and wake analyses have also been studied within the aircraft industry, using control volume analyses of energy, for instance by Denton [11], Drela [12] and Capitao-Patrao et al. [13]. Designers developing novel aircraft concepts, such as Boundary Layer Ingestion (BLI), are actually facing very similar design issues as ship propulsion system designers, with propulsion units operating in the wake of the craft, where the counteracting forces of thrust and drag cannot be studied separately.

Based on an evaluation of the suggested methodologies, it was decided to focus on the use of a control volume analysis of energy, similar to what also has been suggested by Schuiling and van Terwisga [9, 10], Capitao-Patrao et al. [13], and Drela [12]. The summary of the evaluation is available as an internal report [14]. The choice to focus on control volume analyses of energy is motivated by that it is desirable to have an analysis methodology and measures which can be directly related to the delivered power. 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. On the other hand, 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 us with a clear relationship to the ship power consumption.

Further, it was decided to evaluate the viscous losses as internal and turbulent kinetic energy, similar to how the method is applied for compressible flow within the aircraft industry. This enables us to easily visualize the viscous losses, as well as allowing for flexibility in grid generation and simplifying the post processing. Since the internal

energy is obtained through $\hat{u} = c_p T$ (c_p = specific heat capacity, T = temperature), a temperature field is required, implying that the energy equation needs to be solved for in the CFD.

In Section 2.1 it will be described how Reynolds Transport Theorm can be applied to express the delivered power as energy fluxes and pressure work over the control volume surface, a summary of the descriptions available in Paper II-IV. Thereafter, in Section 2.2, aspects to consider when selecting the control volume will be discussed. In Section 2.3 requirements for the underlaying CFD solution are listed, and finally in Section 2.4 benefits and limitations of the presented analysis methodology are summarized. For examples of application of the control volume analysis of energy, the reader is referred to Paper II-IV.

2.1 Reynolds Transport Theorem of Energy Applied on a Marine Propulsion System

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.* The control volume could be of arbitrary shape, which is of importance to facilitate analyses of various kind of propulsion systems. Figure 2.1 illustrates a possible control volume surrounding skeg, propeller and rudder. The control volume is bounded by both the virtual control volume surface, as well as the material surfaces, e.g. 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. The establishment of a suitable control volume for the analyses will be discussed further in Section 2.2.

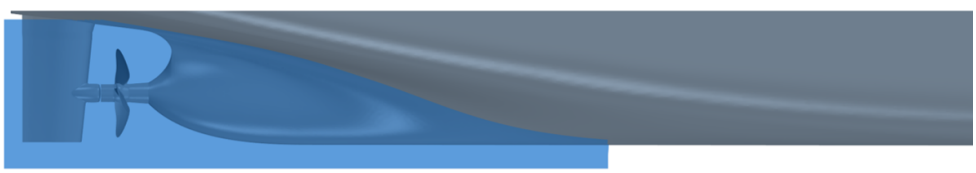

Figure 2.1: Control volume enclosing skeg, propeller and rudder.

The analysis is based on the energy conservation equation, which reads [15],

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

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.

In general, the flow around a marine propulsion system is periodical or fully unsteady. This is tackled through conducting energy flux balances at several time instances, for example at each time step, over a certain period of time. For a periodic unsteady flow the energy flux balance needs to be evaluated over at least one period (note that this was not conducted in Paper III, but is highly recommended). To obtain a representative energy flux balance, averaging of all energy flux balances are conducted. For the control volume analysis, at each time instance it is assumed that the flow is steady, which simplifies the equations considerably and does not introduce especially large errors since the energy flux balance is obtained through the averaging describe above.

Denoting energy per unit mass with e, the energy conservation equation without heat transfer using the Reynolds Transport Theorem for stationary flow yields [15],

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

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

$$P_D = 2\pi n M, \tag{2.4}$$

where M is the torque evaluated over all rotating material surfaces in CS and n denotes 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 volume 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 volume surface or if they are placed outside the boundary layer, shear stresses are expected to be lower. For all cases studied in Paper II-IV, it have 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:

$$e = \frac{1}{2}V_x^2 + \frac{1}{2}(V_t^2 + V_r^2) + \hat{u} + k, \tag{2.5}$$

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, i.e. the direction useful thrust is generated in. This decomposition has been applied by 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. 2.5 and the above mentioned decomposition of the work rate into Eq. 2.2, we obtain:

$$P_D = \int_{CS} (\frac{p}{\rho} + \frac{1}{2}V_x^2 + \frac{1}{2}(V_t^2 + V_r^2) + \hat{u} + k)(\overrightarrow{V} \cdot \overrightarrow{n})dA + \dot{W}_{v,virtual}. \tag{2.6}$$

This equation shows that it is possible to express the delivered power, which traditionally is evaluated through the forces acting on the propeller and its rotation rate, as a sum of energy fluxes and rate of pressure work over the surfaces forming the control volume. Below the different terms which appear in Eq. 2.6 are discussed in more detail.

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 volume boundaries. 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, focusing on these terms from a propulsor 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 [12]. For a propeller in open water, consider a control volume enclosing the propeller, the upstream boundary 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 boundaries must be streamlines where $p = p_\infty$. For such a control volume the evaluation can be limited to the control volume downstream boundary (*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,

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

Through multiplication of all momentum flux balance terms with the advance velocity, the thrust power is obtained,

$$P_x = F_x V_A = \int_{out} (V_A \Delta p + \rho V_A^2 \Delta V_x + \rho V_A (\Delta V_x)^2) dA. \tag{2.8}$$

Performing a control volume analysis of energy for the same control volume gives us the required delivered power expressed as a sum of energy fluxes (similar to Eq. 2.6),

$$\begin{aligned} P_D = \int_{out} (\Delta p \Delta V_x + V_A \Delta p + \rho V_A^2 \Delta V_x + \rho V_A (\Delta V_x)^2 + \\ \frac{1}{2}\rho (\Delta V_x)^2 V_x) dA + \int_{out} (\frac{1}{2}(V_t^2 + V_r^2) + \Delta \hat{u} + \Delta k) \rho V_x dA. \end{aligned} \tag{2.9}$$

$\Delta \hat{u}$ 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. 2.9), the thrust power (Eq. 2.8) can be identified as well as two additional terms, denoted the secondary axial kinetic energy flux,

$$\int_{out} \frac{1}{2}\rho (\Delta V_x)^2 V_x dA, \tag{2.10}$$

and the pressure defect work rate,

$$\int_{out} \Delta p \Delta V_x dA. \tag{2.11}$$

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.

Note that the evaluation of secondary axial kinetic energy flux and pressure defect work rate is not possible for the general control volume enclosing a propeller in open water, since the lateral boundaries are not streamlines with $p = p_\infty$ and the inlet conditions are not required to be homogeneous. For a propeller operating in open water applying an arbitrary control volume with boundaries in the vicinity of the propeller, the axial non-uniformity losses can however be estimated indirectly. This estimation is obtained through using the difference between the sum of axial kinetic energy flux plus rate of pressure work and the thrust power evaluated from forces acting on the propeller. This

way of evaluating the axial non-uniformity losses will always underestimate them to some extent, since a part of the spatial wake non-uniformity already will be mixed out and converted to internal energy.

If instead focusing on the complete marine propulsion system, which actually is what we really are interested in, the combined sum of rate of pressure work and axial kinetic energy flux should be viewed upon as entirely non useful energy fluxes, i.e. axial non-uniformity losses, from a ship point of view. This is due to that in the ideal case, the propeller's slipstream, which ideally 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 2.2. A propeller operating in open water conditions can never obtain zero axial non-uniformity losses, even if it is an actuator disk, since there always will be losses due to the velocity gradients present between the propeller slipstream and surrounding flow, causing downstream mixing losses.

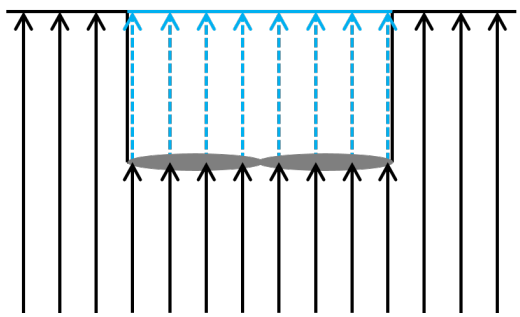

Figure 2.2: Sketch of an actuator disk completely filling a wake of a vessel, illustrating a case with zero axial non-uniformity losses.

In summary, for a propulsion unit operating in open water the axial kinetic energy flux and rate of pressure work consist of both useful thrust generation as well as axial non-uniformity losses, but for a ship with propulsion system the combined sum of axial kinetic energy flux and rate of pressure work, should be viewed upon as entirely axial non-uniformity losses.

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 unit or vessel 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. In case the transverse kinetic energy outflow of the control volume is reduced in comparison to that of the control volume inflow, for instance by means of a rudder, this term will become negative which may indicate the recovery

of the unfavorable transversal components to useful energy components.

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, where turbulence 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 the size of wetted surfaces. 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.

The internal energy is obtained through $\hat{u} = c_p T$, i.e. a temperature field is required from CFD, implying that the energy equation needs to be solved.

2.2 Establishment of Control Volume

In the establishment of a control volume there are several aspects to consider, two important are accuracy and possibility to characterize the system performance.

With accuracy, the accordance between left and right hand side of Eq. 2.6 is considered, i.e. the accordance between the power evaluated through the forces acting on the propeller and its rotation rate, and the sum of all energy fluxes. For a propeller operating in open water, as reported in Paper II, the obtained difference between the two terms was less than 1 %. Slightly larger difference between terms are observed for control volumes surrounding marine propulsion systems operating behind ships, but still less than 3 %, as noted in Paper IV. The evaluation of the energy fluxes does not include the work performed by shear stresses on the virtual control volume surfaces. However, both for a propeller in open water (Paper II) and complete marine propulsion system studies (Paper III), they have been negligible, constituting less than 0.01 % of the delivered power. The discrepancies are instead assumed to be caused by the effect of numerical dissipation, numerical convergence and inaccuracies of evaluating energy fluxes over the control volume. To minimize these errors it is recommended to place the control volume within a region with relatively fine grid and to avoid regions with strong gradients if possible. We have used different control volume shapes, both cylindrical and rectangular boxes, in Paper II-IV. The box-shaped control volume was motivated by that the interpolation errors could be reduced, due to that it could be aligned with the same coordinate system as the grid.

Of high importance when establishing the control volume is also the possibility to characterize the system performance. It will be highly beneficial for the analysis if the downstream control volume surface is placed in the vicinity of the propulsion system. When moving away from the system the share of internal and turbulent kinetic energy

fluxes increase, due to viscous and turbulent dissipation associated with mixing out of spatial non-uniformities in the flow. Obviously more information about the flow is obtained in the vicinity of the propulsion unit, whereas further away the kinetic energy terms are to a larger extent converted to internal and turbulent kinetic energy. The control volume extension upstream also plays an important role in how one can distinguish and interpret the beneficial energy components from the unfavorable ones. From a complete ship perspective, as describe in Section 2.1 above, the sum of axial kinetic energy flux and rate of pressure work should be viewed upon as entirely axial non-uniformity losses. On the other hand, if the control volume only encloses a certain domain of the ship, such as the aft ship, 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 rate of pressure work and axial kinetic energy flux terms must be useful thrust power, similar to a propeller operating in open water. Such a control volume could still be beneficial due to other reasons, for instance to only enclose the domain of interest can facilitate more focused studies. Figure 2.3 illustrates control volumes with different extension as applied within Paper III.

Figure 2.3: Alternative control volumes enclosing: the propeller and duct, the aft-ship and the entire vessel.

The control volume analyses conducted so far does not enclose a free surface. It is however possible to also include a free surface in the control volume. Compared to a double-body model the vessel resistance will be higher, for the control volume analyses it will imply more kinetic energy losses and viscous losses in the vicinity of the surface. This might be important if the propulsion system interacts with the free surface.

For the analyses conducted in Paper III and IV we considered it beneficial to also split the control volume into separate parts, covering the skeg, propeller and rudder respectively, to be able to analyze the difference between systems in more detail, see Figure 2.4 for an illustration of such control volumes. A control volume only enclosing the hub cap was also constructed to study the hub vortex specifically. There is actually

no limits of how to construct a control volume, but should rather be seen as a step in exploring the details of the flow around the propulsion system.

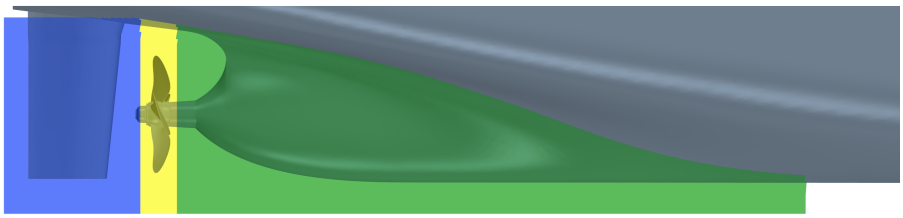

Figure 2.4: Control volume enclosing the aft-ship split into three internal control volumes.

Finally, an important remark, the distribution into the different energy fluxes will be highly dependent on the location of the control volume. It is therefore necessary to apply identical control volumes when comparing the energy fluxes between different designs. Important to note is also that the energy flux balance analysis will only describe the performance within the constructed control volume, implying that the control volume has to enclose the entire flow domain of interest.

2.3 Requirements on the CFD solution

For repeatability and an efficient working procedure, Eq. 2.6 is preferably implemented in a post-processing script. To conduct a control volume analysis of energy, access to commonly available variables is required, and to be able to set it up as a post-processing procedure only, the possibility to evaluate the variables on surfaces constructed in the post-processing phase is necessary. To establish the control volumes as a post-processing step is highly recommended, compared to using predefined surfaces or interfaces, since it enables higher degree of flexibility when analyzing the results. Using STAR-CCM+ it means that the control volumes are constructed with the aid of extracted surfaces (derived parts).

The CFD solution requires a solved temperature field, to be able to evaluate the internal energy flux. Due to that the temperature increase caused by viscous and turbulent dissipation is very low it requires a well converged temperature field with high precision of the values to not introduce large errors in the evaluated internal energy flux.

Within this project the control volume analysis is only implemented in STAR-CCM+, but it should be possible to implement it for any CFD software with those basic functionalities mentioned above.

Grid

As shown in Paper II the distribution into the different energy flux components can be dependent on the grid resolution. In general a more refined grid implies a slightly reduced contribution from internal energy flux, which most probably is related to that a finer grid

implies a reduced numerical dissipation. For Grid 2 in Paper II, which can be considered representative for the grids applied also in Paper III and IV, the amount of numerical dissipation contributing to the internal energy flux is estimated to 6.5% of the total internal energy flux. For comparative studies between different designs it is therefore strongly recommend to keep a similar grid resolution between the cases. Furthermore, the same will hold as always for CFD analyzes, the grid refinement determine the flow details possible to predict, so will it also determine which flow details you can study through a control volume analysis.

2.4 Using the Energy Fluxes to Characterize and Improve Systems

The delivered power has been expressed through a sum of energy fluxes over the control volume surface in Paper II, III and IV. In Paper III bar charts and tables clearly shows that the additional required power for the ducted propeller configuration to the largest extent is due to the system being associated with higher viscous losses, i.e. internal and turbulent kinetic energy fluxes. In Paper IV the optimal propeller diameter in open water is explained using efficiency curves with decomposed losses, see Figure 2.5. The optimal diameter is described as a trade-off between blade load/flow acceleration, represented by transverse kinetic energy and axial non-uniformity losses, and viscous losses. A small diameter would imply higher load on each blade section and losses associated with that, while a too large propeller costs more in terms of viscous losses. This trade-off between the different losses is valid also for analyses of complete propulsion systems, such as the optimal propeller diameter in behind conditions, also studied in Paper IV. In other words, reducing one loss component will most probably increase another, and where lies the most optimal solution? That is really the question. Important to point out in association to this, is that this trade-off, most probably not only is highly dependent on operating conditions, but also on scale. Viscous losses are in general much lower in ship scale due to the higher Reynolds number, which indicates that the 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. Figure 2.6 illustrates the efficiency curve for a propeller in model and ship scale respectively, the reduced contribution from viscous losses in ship scale is substantial.

From Paper III and IV it is clear that the decomposition into different energy fluxes can give us a main characterization of the system in relation to other systems. However, in many cases we are interested in a more detailed description, to be able to deduce which flow features the energy fluxes are linked to. One part, of such more detailed studies, could be to form several control volumes enclosing different parts of the system to be able to quantify the performance of each part of the system, as mentioned in Section 2.2 and conducted in Paper III and IV. Naturally if the control volume does not enclose any propulsor surfaces, the total sum should equal zero and not P_D. For such a control volume it is however still interesting to study the energy conversion process. A good example is the control volume surrounding the rudder in Paper III, for both configurations the main energy conversion process is from pressure work and transverse kinetic energy to axial

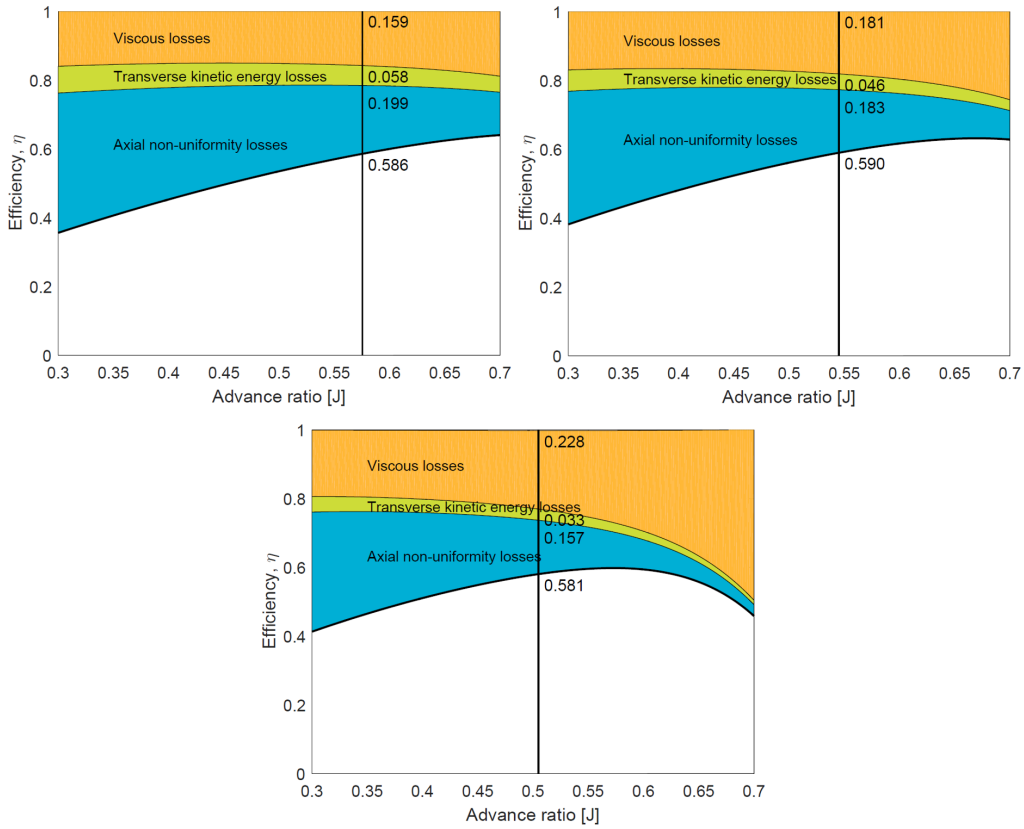

Figure 2.5: Open water efficiency versus advance ratio for model scale propellers. Top left: $D_P = 3.8m$, top right: $D_P = 4.0m$, bottom: $D_P = 4.3m$. Area between efficiency curve and unity decomposed into different hydrodynamic losses. The size of each component at studied design operating point (marked with black line) printed in figure.

kinetic energy and internal energy. For the open propeller much energy is converted to axial kinetic energy, while the figure for the ducted propeller configuration is significantly lower, with a much larger share of conversion to internal energy flux, i.e. energy turned into viscous losses.

For further understanding, and to be able to pinpoint parts to be addressed for design improvements, visualizations of the flow field are strongly recommended. Visualizations of CFD results, through for instances animations, contour plots or isosurfaces, are absolutely no new invention. But the knowledge about losses that the control volume analysis provide us with, can certainly assist when studying the results. For instance knowledge about all velocity perturbations downstream the system, both positive and negative, relative to the vessel speed, indicating the presence of axial non-uniformity losses, can help us. Another side effect from the control volume analysis of energy is the solved temperature field, which can be very useful. The temperature field or internal energy flux can be used

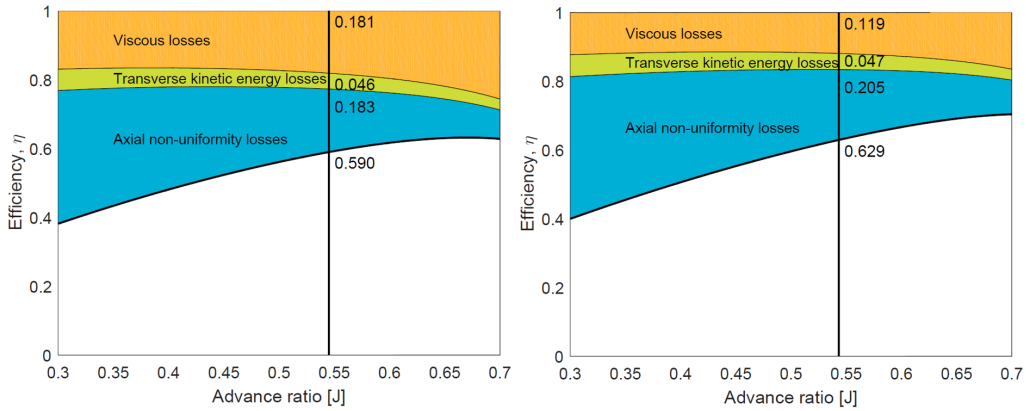

Figure 2.6: Open water efficiency versus advance ratio for propeller with $D_P = 4.0m$. Left: Model Scale, right: Ship Scale. Area between efficiency curve and unity decomposed into different hydrodynamic losses. The size of each component at studied design operating point (marked with black line) printed in figure.

in several different ways to clearly illustrate the viscous losses, which can be addressed for design-improvements. For instance, in Paper III it was clearly seen 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. Could these be removed to reduce the viscous losses, or would it cost too much in terms of increased transverse kinetic energy losses or reduced maneuverability capacity? We also know that the losses over the blades are strongly related to the propeller inflow. The analysis is therefore preferably complemented with a presentation of the effective angle of attack in behind conditions. Needed for that is beside the flow field, the propeller rotation rate and metal angles at the propeller leading edge.

Concluding this, it is clear that the tool is not absolute, in the sense that the decomposition into different energy fluxes is independent of user. It is rather an outcome of a combination of CFD modelling, grid and especially choice of control volume. But established as a standard procedure within a organization, it will absolutely be possible to conduct comparative analyses. Due to this lack of general characterization possibilities it will not be a tool replacing the classical interaction effects approach. To create a tool that will be absolute and independent of user, we would most probably need to go for one which is based on forces on the ship, including rudder, propeller and other appendages, and not on the flow field, at least for the nearest future. This is motivated by that the differences in the flow generally are much larger between different CFD modeling approaches compared to the differences on ship surfaces and it would also be a challenge to establish general locations of where to evaluate the flow. But the question is whether a tool based on surface forces really could describe the hydrodynamics of a system? Using CFD, the surface forces can be decomposed on a more detailed level compared to the classical interaction effects. This was for instance conducted in Paper IV, where the rudder and hull forces were presented separately, however, this did not provide us with a clear

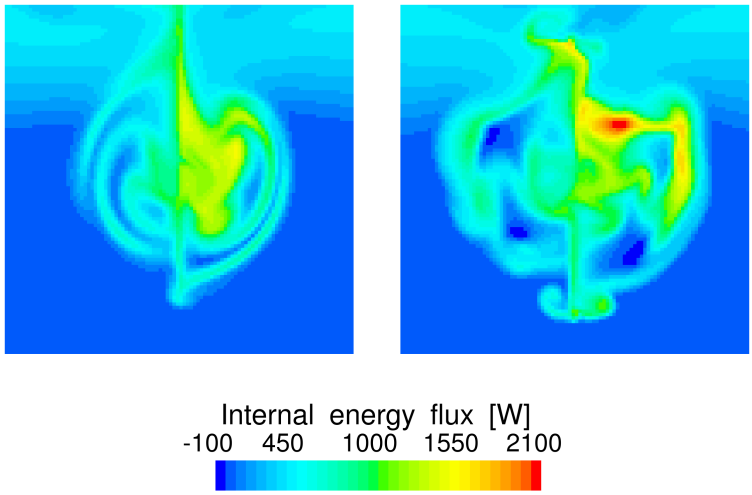

Figure 2.7: Contour plot of internal energy flux downstream rudder from study conducted in Paper III. Left: open propeller configuration, right: ducted propeller configuration.

description of the propulsion system performance within that study. However, it is still considered that using energy flux balances and associated post-processing tools, we have taken some clear steps to better understand and possibility to improve the hydrodynamic performance of different propulsion systems.

3 Simulating the Flow around Marine Propulsion Systems

This section aims to conclude some important findings and aspects considered during the project leading up to this thesis, and does not contain a full description of how to perform CFD simulations for marine propulsion systems. For more complete recommendations for ship RANS simulations in model scale the reader is referred to the ITTC guidelines [16, 17] and the summary of the 2010 workshop in ship hydrodynamics [18]. For specific details on propeller modeling, the results from the validation workshop on the Potsdam propeller test case (PPTC) at the Symposium on Marine Propulsors in 2011 and 2015 are recommended [19, 20].

Best-practice guidelines for CFD in ship scale are not really available yet, a natural consequence of the lack of relevant data to use for validation. A workshop on ship scale hydrodynamics was held in 2016 [21], but it is difficult to set out any clear best-practice guidelines only based on this workshop. Hopefully, more ship scale flow data is collected in the near future, which could be used for validation of CFD methods.

The CFD software used for the work presented in this thesis is STAR-CCM+, therefore some recommendations may be specific for this software. The versions of STAR-CCM+ used within the project varies from v10.04 to v12.06. It is worth to note that commercial CFD software are continuously developing with new models incorporated or improvement of existing ones, which also implies that the findings and aspects brought up below, may become outdated.

3.1 Grid Generation

For studies with focus on the propulsion system it is important with a refined grid around the aft part of the vessel to capture the thick boundary layers constituting the propeller inflow. A refinement region with cell sizes of about $0.015D_P$, extending 5-10 D_P upstream the propulsion unit has generally been applied. Beside this it is important to capture the flow details of the boundary layer all the way to the bow, using prism layers and sufficiently refined grid, to ensure a representative propeller inflow. For more general recommendations on grid construction and grid refinement, see for instance [18, 19, 20].

An important question for the grid generation is the use of wall function grids or resolving the boundary layers down to the wall. This cannot really be discussed without considering the models to be used, and will therefore be discussed in conjunction with turbulence models in Section 3.3.

3.2 Modelling Propeller Rotation

Common methods to model propeller rotation includes: various virtual propellers using source terms to the momentum equations; stationary resolved propellers using multiple reference frames (MRF); and rotating propellers utilizing sliding grid interfaces.

In the studies presented in this thesis only resolved rotating propellers have been used, rotating 1° per timestep. Out of the three alternatives mentioned, it is the most representative model for the real flow conditions, but also by far the most computationally costly one. Through a resolved rotating propeller, it is possible to capture the increased losses or gains caused by a modified propeller inflow in behind conditions or changed operating conditions. For the studies conducted in Paper III and IV it was not an alternative to use a resolved stationary propeller using MRF, due to the large variations in propeller inflow. A stationary propeller implies that the propeller performance is entirely dependent on its fixed position in the wake. Variations in delivered power and thrust of $\pm$ 5 % or more are not uncommon dependent on position. A stationary resolved propeller using MRF should therefore only be considered for relatively homogeneous inflow conditions. Since also the propeller outflow will be dependent on the position of the propeller, it would further not be suitable for a configuration including rudder. An alternative for homogeneous inflow together with rudder could however be to apply mixing plane interfaces, common within other turbo machinery applications.

A virtual propeller model is more able to represent the function of a propeller in a inhomogeneous wake, in relation to a stationary resolved propeller using MRF. However, it is not capable of describing the flow in detail since it does not include the blades. It will neither be possible to capture the viscous losses around the blades, which implies that they cannot be represented in a control volume analysis of energy. In other words, the control volume analysis of energy as described in Section 2 is not really applicable for analyses utilizing virtual propeller models. Furthermore, virtual propeller models are often dependent on provided data based on operation in open water, not always relevant for the operating conditions in behind, with varying axial, radial and tangential flows around a revolution and radially along the blade. This also make them less suitable for more detailed studies of the propulsion system performance, since all interaction effects may not be properly captured.

3.3 Modelling Turbulence

Turbulence modelling is crucial for a correct representation of the boundary layers constituting the inflow to the propulsion unit. It is especially critical in model scale where the boundary layers are thicker, due to the lower Reynolds number, compared to ship scale. These 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.

Suitable turbulence models for the investigation of ship hydrodynamics are thoroughly discussed in [18]. The main conclusion is 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).

For our studies on the Japan Bulk Carrier (JBC) test case, reported in Paper I, both k-ω SST with curvature correction and RSM provide accurate predictions of the wake field on grids resolving the boundary layers, see Figure 3.1. However, k-ω SST with

resolved boundary layers results in a severely underestimated hull resistance and RSM is computationally expensive to use. The computational cost of RSM is partly due to the larger number of equations which needs to be solved, but also due to that highly anisotropic cells, which preferably are used for free surface simulations to reduce cell count, may need to be avoided, since they often result in convergence problems for RSM. These evaluations were conducted using STAR-CCM+ v10.06; since then new turbulence models, such as EASM, have been released in STAR-CCM+. More recently, several turbulence models have been evaluated using STAR-CCM+ v12.06. We are however still not convinced about the most optimal model, providing accurate predictions for both resistance and wake field, and also preferably robust and computationally efficient. RSM is most probably the best choice in terms of accuracy, but it is known to be difficult to converge, and therefore not really a good alternative for standardized CFD setups, which preferably are automated.

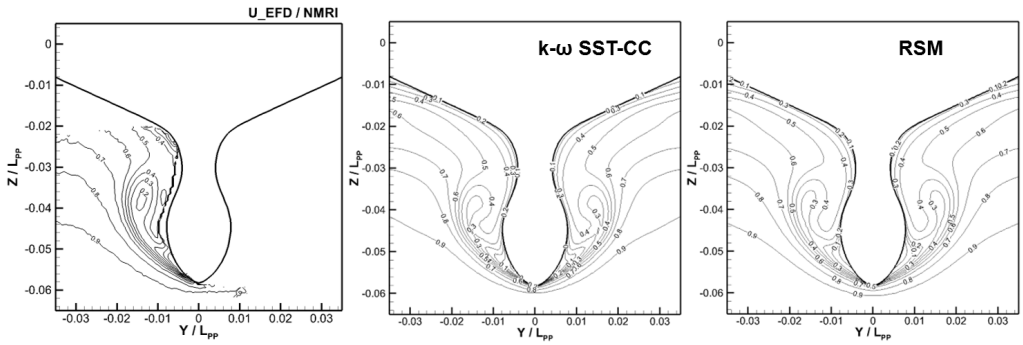

Figure 3.1: Mean axial velocity at $x/L_{pp} = 0.9625$. Double-body model CFD results compared to measured data (left).

For the studies in Paper III and IV, k-ω SST with curvature corrections using wall functions was applied. For the geometry studied in Paper III we had access to a measured bare hull wake field, and for this case the computed bare hull wake field using k-ω SST with curvature correction together with wall functions was in much better agreement with the measured wake field, compared to for the JBC test case. However, this combination, k-ω SST with curvature corrections using wall functions, cannot be recommended generally due to a poor performance on the JBC test case, see Paper I.

Until now, in principle all validation of turbulence models for ship hydrodynamics are conducted in model scale, due to lack of measured flow data on ships. It is assumed that the turbulence model will be less critical for representative predictions of the boundary layers in ship scale, due to the higher Reynolds number and associated thinner boundary layers.

3.4 Modelling Laminar to Turbulent Transition

For propellers in model scale, both in open water and in behind conditions, the turbulence models are challenged by the low Reynolds numbers (chord-based) often implying laminar flow over a part of the blade, influencing thrust and torque characteristics. Models predicting laminar to turbulent transition are available, such as the one by Langtry and Menter [22, 23]. A good description of how to apply the $\gamma - Re_{\Theta}$ transition model for marine propellers in open water is provided by Bhattacharyya et al. [24]. However, since the transition models most often are relatively sensitive to different factors, they could be more troublesome to apply for complicated cases, such as propellers operating behind the ship.

The computational cost associated with a transition model, beside the solution of additional equations, is linked to requirements of refined grids. To be able to capture the laminar to turbulent transition, grid refinements in the transverse directions of the boundary layers are generally required, as well as a low expansion ratio between the prism layers. In Paper II the required grid refinements for a transition model implied 50 % increase in the propeller domain cell count, in relation to a standard grid. This additional computational cost is not a big issue for propeller open water CFD, but for self-propulsion CFD, with transient flow, free surface and sliding mesh interfaces, it will increase the computational time substantially.

The issue with laminar flows over a part of the blades is just as important in behind conditions, where the rotation rate, and consequently Reynolds numbers, often are even lower than for corresponding open water tests. A complicating factor for self-propulsion CFD is that the transition model also will affect the flow over the hull. In model tests the boundary layers are most often triggered to turbulent flow just behind the bow, in other words you do not want a prediction of the boundary layer as it would have appeared without triggering. Further, the transition models are normally sensitive to the turbulence intensity of incoming flow. The level of turbulence intensity often decays in an unrealistic high pace for CFD, from the inlet boundary into the domain. Different measures to counteract this decay and obtain more realistic values just upstream the propeller are therefore most commonly applied for propellers in open water, such as described in [24]. For a propeller operating behind a ship the turbulence intensity will naturally be high for the most intense part of the wake, also in CFD, however for the region outside of this, there is a risk for too low turbulence intensity, also in behind conditions. It will be more complicated to apply any measures to counteract this for a self-propulsion CFD, due to the risk of influencing the developed flow field around the hull in a non-physical manner.

The influence of partly laminar flows on the propeller in self propulsion is something that needs to be studied further. Fortunately, this is a problem limited to model scale and not relevant for the real systems.

3.5 Modelling Surface 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. However, in ship scale the surfaces are 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 unit inflow. A good illustration of the effect of applying different hull roughness in CFD on the wake is provided in [25].

The standard hull roughness measured on ship hulls is only a measure of the height of the surface roughness [26], and does not contain any information about its other characteristics. Research to characterize the hull roughness due to paint and organic growth has been conducted, for instance by Schultz [27]. The influence by other surface imperfections are obviously more difficult to characterize in a general manner since they are dependent on the location of the imperfections as well as the hull form.

Common roughness functions, implemented in commercial CFD software, are based on equivalent sand grain roughness. Schultz [27] suggested to use 17 % of the measured hull roughness as an equivalent sand grain roughness, based on his experiments and using a Colebrook-type roughness function. However, the question is also if the common roughness functions developed based on equivalent sand grain roughness is representative for the hull roughness. Alternative roughness functions, adapted to roughness representative for hulls, have been developed by Demirel et al. [28, 29]. It must however be kept in mind that all these results are dependent on the characteristics of the hull roughness within those specific studies.

Another important question for a specific CFD study, is to determine which roughness that should be considered as representative, the one of a clean painted hull or the one of a ship in service? It may seem more natural to use the one for the average fouling of a ship in service, however a clean painted hull is more representative for the sea trial conditions and also what is aimed for in the ITTC-78 performance prediction method [30, 31].

In Paper IV, a backward-engineering approach was made to represent the hull roughness. We decided to conduct the study for a clean painted hull and use the standard hull roughness according to ITTC-78, 150 μm. Applying the guideline that a corresponding equivalent sand grain roughness should be 17 % of this value, implied a very low resistance increase on a bare hull, most probably due to the use of a slightly different roughness function in STAR-CCM+ compared to the one applied in [27]. We therefore decided to aim for the resistance increase obtained using the roughness allowance formula in ITTC-78 prediction method [30], which for the studied case was 12-13 % for a bare hull, and then adjust the equivalent sand grain roughness through an iterative procedure. Through bare hull CFD simulations for this case, with smooth and rough surfaces, it was found out that 80 μm was associated with a 12.7 % resistance increase compared to a smooth hull.

For ship scale CFD it could also be considered to apply a surface roughness on the propeller. Such a roughness is an important factor for the ITTC open water characteristics scaling method [30], as for instance is shown in Paper IV. However, the standard rough-

ness recommended in ITTC-78 [30] of 30 μm is 10 times larger than the ISO requirement (ISO 484/1:2015 Class I) of the manufactured propeller surface roughness to be less than 3 μm (the details of how this roughness is defined is included in the ISO standard). The ISO requirement of 3 μm for most cases could be considered as a hydraulically smooth surface, while the ITTC-78 recommendation of 30 μm cannot. Also for the propeller it is therefore relevant to consider which state of the propeller surface that is most relevant to consider in a specific study.

3.6 Modelling the Free Surface

The propulsion system most often interacts with the water surface and wave system and the CFD simulation therefore needs to include modelling of the free surface. This poses several challenges and increases the computation cost. For the study conducted in Paper IV, where the differences in power between the different propeller diameters were only about 0.1 %, it would have been very challenging to obtain the accuracy required with a free surface, due to the transient flow features often occurring in the vicinity of the surface.

The recommended and commonly applied model for the free surface using STAR-CCM+ is the Volume-of-fluid (VOF) method [32], implying that the domain consists of one fluid whose properties vary according to the volume fraction of water/air. The convective term is discretized using the High Resolution Interface Capturing (HRIC) scheme [33]. A common issue with this model is that air from the surface is transported down along the hull surface, generating domains around the hull with the fluid being a mixture of air and water. This modelling error influences the resistance of the vessel as well as the inflow to the propulsion unit. Possible alternatives to avoid this is reduction of the time step to limit the CFL number, which introduces another additional computational cost, or artificial measures removing the air content on specified locations. Introduction of artificial measures, may naturally also affect the modelling of other phenomena in an undesirable manner. On the other hand, if the other alternative is to not include the free surface at all, even larger errors may be introduced due to both upstream and downstream influences of a symmetry plane or fixed surface on the propulsion system performance.

4 Summary of Papers

4.1 Paper I

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

Motivation and Divison of work

As a foundation for further studies on the hydrodynamic performance of marine propulsion systems a CFD validation study was conducted using the JBC test case for the bare hull and propeller in open water separately.

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 conducted the CFD validation study, i.e. generated the grids, set up the simulations, post processed and analyzed the results with support from the other authors.

Results and Conclusion

For the bare hull we had access to measured resistance, sinkage, trim, detailed wave profiles and wake field at three axial positions, which provided us with good possibilities to conduct a CFD validation study. A mesh sensitivity study combined with an evaluation of different turbulence models showed us that wall functions were not sufficient to accurately predict the wake field. Using grids with resolved boundary layers, k-ω SST with curvature correction and RSM provided accurate predictions of the wake field. However, k-ω SST with resolved boundary layers resulted in a severely under estimated hull resistance, and using RSM we did not manage to achieve a converged solution with a free surface. This implied that we did not manage to establish a method that both provides an accurate prediction of the stern wake as well as the resistance, except from using RSM, which is both time consuming and requires a very high quality grid (note that it is possible to obtain a converged solution, even if we did not manage within this study).

For the propeller in open water several grid sensitivity studies were conducted, but despite this the accordance with measured test data was slightly worse than expected. One possible reason behind the differences could be that the CFD model wrongly assumes the propeller boundary layer to be fully turbulent, which may not be the case for the tested configuration (Re = $4{\cdot}10^5$).

4.2 Paper II

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 Divison of work

The need for new ways of analyzing and characterizing the hydrodynamic performance of marine propulsion system has been well motivated in Section 1. This paper describes a control volume analysis of energy applied on a propeller operating in open water. The aim with this simplified case was 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 delivered power can be expressed with high accuracy using the energy fluxes through the control volume surface, the difference compared to the power evaluated based on integrated forces on the propeller surface was less than 1% within this study. The division between the different energy fluxes is found out to be slightly 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.

4.3 Paper III

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 Divison of work

This paper describes a control volume analysis of energy applied on a model-scale cargo vessel equipped with an open and ducted propeller configuration, respectively. The motivation behind the paper was to show that the energy balance analysis can be applied on a complete propulsion system, and show that it can be a useful tool when analyzing the interaction effects.

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 generated the grids, set up the simulations, post processed and analyzed the results and wrote the paper.

Results and Conclusion

The ducted propeller configuration has a much higher required delivered power compared to the open propeller. Through the energy balance 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.

4.4 Paper IV

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

Motivation and Divison of work

In the preliminary design of a propulsion unit the selection of propeller diameter is most commonly based on open water tests of systematic propeller series. The optimum diameter obtained from the tested propeller series data is however 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. Rickard E. Bensow and Arash Eslamdoost contributed with their ideas in how to present the results and structure the 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. Trying to understand the reasons behind this reduction in optimal propeller diameter in behind conditions, energy flux balances were applied. The 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. deflect the flow tangentially to a larger extent, compared to a larger propeller delivering the same power. Generally, for it to be optimal with a smaller diameter in behind in relation to open water, it requires that the gain in transverse kinetic energy losses thanks to the rudder overcomes the increase in viscous losses, which the complete system, including rudder, inevitably causes.

5 Concluding Remarks

Analyzing the performance of propulsion systems using control volume analysis of energy can provide us with many useful insights in propulsion system interaction phenomena. It is also pointed out that the control volume analysis most preferably is complemented with a detailed study of the flow field to be able to pinpoint possible improvements of the systems. In relation to the traditional measures for interaction effects, the key difference is really the focus on the flow field instead of on forces on the material surfaces to understand the functioning of the system.

Some important aspects to consider when simulating the flow around marine propulsion systems are summarized in Section 3. It is of highest importance that the models building up the complete CFD-model is representative for the real flow to be able to draw conclusions about the performance of the propulsion system. Bare hull and propeller open water CFD have reached a high level of maturity, with the aid of considerable validation work. Despite this, I would claim that self-propulsion CFD with focus on a detailed representation of the propulsion system, is not really there yet. Partly it may be due to the additional cost associated with self-propulsion simulations, implying that simplifications often are carried out, such as simplified propeller models. Another contributing factor may be that the measurements for these kind of tests most often are less detailed and on a more overall level. A third contributing factor is for sure that many parties cannot access all the detailed geometries necessary to set up a full self-propulsion CFD model.

As a recommendation for future work, to create a useful tool for propulsion system design work, I think that it is of highest importance that the control volume analysis of energy is further adapted by the industry, and used in daily work, to suit their own needs. From an academic point of view, it would be interesting to continue to focus on application of the control volume analysis of energy to illustrate and explain more common but poorly understood propulsion system interaction phenomena, such as the optimal propeller diameter in behind study (Paper IV). It would also be interesting to study a commonly installed Energy Saving Device (ESD) in both model and ship scale, to be able to explain the differences in performance between the scales using a control volume analysis of energy.

For future CFD validation work, more efforts are recommended to be spent on self-propulsion CFD and detailed propulsion system modelling, in both model and ship scale. Important issues to focus on is how to model the hull roughness, suitable choice of turbulence model, and the possible influence by laminar flow over the propeller blades in model scale. For ship scale, more flow data which could be used for validation of CFD methods would be warmly welcomed.

However, independent of further developed methods to conduct CFD simulations of the propulsion system and establishment of methods to analyze these results, I believe that the single most important factor to obtain more effective propulsion systems is to open up for more system design possibilities. In other words, removing the limitations caused by the markets requirements on confidentiality for their designs, at least within certain collaborations or for certain partners.

References

[1] 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.

[2] 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.

[3] 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.

[4] 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.

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

[6] 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.

[7] 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.

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

[9] 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.

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

[11] J. D. Denton. The 1993 IGTI Scholar Lecture Loss Mechanisms in Turbomachines. *Journal of Turbomachinery* **115** (1993), 621–656.

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

[13] A. Capitao Patrao, R. Avellán, A. Lundbladh, and T. Grönstedt. "Wake and Loss Analysis for a Double Bladed Swept Propeller". *Proceedings of ASME Turbo Expo 2016*. Seoul, South Korea, 2016.

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

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

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

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

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

[19] 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.

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

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

[22] 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 (2004), 413–422.

[23] 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.

[24] 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.

[25] T. Guiard, S. Leonard, and F. Mewis. "The Becker Mewis Duct - Challenges in Full-Scale Design and new Developments for Fast Ships". *Proceedings of the Third International Symposium on Marine Propulsors*. Launceston, Tasmania, Austrailia, 2013.

[26] L. Larsson and H. C. Raven. *Principles of Naval Architecture Series - Ship Resistance and Flow*. Jersey City: The Society of Naval Architects and Marine Engineer, 2010.

[27] M. P. Schultz. Frictional Resistance of Antifouling Coating Systems. *Journal of Fluids Engineering* **126**.6 (2004).

[28] Y. K. Demirel, M. Khorasanchi, O. Turan, A. Incecik, and M. P. Schultz. A CFD model for the frictional resistance prediction of antifouling coatings. *Ocean Engineering* **89** (2014), 21–31.

[29] Y. K. Demirel, O. Turan, and A. Incecik. Predicting the effect of biofouling on ship resistance using CFD. *Applied Ocean Research* **62** (2017), 100–118.

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

[31] ITTC. *The Propulsion Commeittee - Final Report and Recommendations to the 28th ITTC*. Tech. rep. Wuxi, China, 2017.

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

[33] S Muzaferija and M Peric. "Computation of free surface flows using interface-tracking and interface-capturing methods". *Nonlinear Water Wave Interaction*. Ed. by O Marenholtz and M Markiewicz. Southampton: WIT Press, 1999.