Revised manuscript (clean version)

Ship-Scale CFD Benchmark Study of a Pre-Swirl Duct on KVLCC2

Jennie Andersson[a,*], Alex Abolfazl Shiri[b], Rickard E. Bensow[a], Jin Yixing[c], Wu Chengsheng[c], Qiu Gengyao[c], Ganbo Deng[d], Patrick Queutey[d], Yan Xing-Kaeding[e], Peter Horn[e], Thomas Lücke[e], Hiroshi Kobayashi[f], Kunihide Ohashi[f], Nobuaki Sakamoto[f], Fan Yang[g], Yuling Gao[g], Björn Windén[h], Max G. Meyerson[i], Kevin J. Maki[i], Stephen Turnock[j], Dominic Hudson[j], Joseph Banks[j], Momchil Terziev[k], Tahsin Tezdogan[l], Florian Vesting[m], Takanori Hino[n], Sofia Werner[b]

[a]*Chalmers University of Technology, Department of Mechanics and Maritime Sciences, 412 96 Göteborg, Sweden*
[b]*SSPA Sweden AB, Chalmers Tvärgata 10, 400 22 Göteborg, Sweden*
[c]*China Ship Scientific Research Center, 214082 Wuxi, China*
[d]*Ecole Centrale de Nantes, LHEEA CNRS UMR 6598, 44321 Nantes, France*
[e]*Hamburg Ship Model Basin, Bramfelder Str. 164, 22305 Hamburg, Germany*
[f]*National Maritime Research Institute, Shinkawa, Mitaka, Tokyo, 181-0004, Japan*
[g]*Shanghai Ship and Shipping Research Institute, State Key Laboratory of Navigation and Safety Technology, 600 Minsheng Road, Shanghai, China*
[h]*SHORTCUt CFD, College Station, Texas, USA*
[i]*University of Michigan, Department of Naval Architecture and Marine Engineering, Ann Arbor, Michigan, USA*
[j]*University of Southampton, Maritime Engineering Group, SO17 1BJ, Southampton, UK*
[k]*University of Strathclyde, Faculty of Engineering, G1 1XW, Glasgow, UK*
[l]*University of Strathclyde, Department of Naval Architecture, Ocean and Marine Engineering, G4 0LZ, Glasgow, UK*
[m]*Volupe AB, Siemens Solution Partner, 436 34 Askim, Sweden*
[n]*Yokohama National University, Faculty of Engineering, Hodogaya-ku, Yokohama 240-0014, Japan*

Abstract

Installing an energy saving device such as a pre-swirl duct (PSD) is a major investment for a ship owner and prior to an order a reliable prediction of

[*]Corresponding author.
Email address: `jennie.andersson@chalmers.se` (Jennie Andersson)

the energy savings is required. Currently there is no standard for how such a prediction is to be carried out, possible alternatives are both model-scale tests in towing tanks with associated scaling procedures, as well as methods based on computational fluid dynamics (CFD). This paper summarizes a CFD benchmark study comparing industrial state-of-the-art ship-scale CFD predictions of the power reduction through installation of a PSD, where the objective was to both obtain an indication on the reliability in this kind of prediction and to gain insight into how the computational procedure affects the results. It is a blind study, the KVLCC2, which the PSD is mounted on, has never been built and hence there is no ship-scale data available. The 10 participants conducted in total 22 different predictions of the power reduction with respect to a baseline case without PSD. The predicted power reductions are both positive and negative, on average 0.4%, with a standard deviation of 1.6%-units, when not considering two predictions based on model-scale CFD and two outliers associated with large uncertainties in the results. Among the variations present in computational procedure, two were found to significantly influence the predictions. First, a geometrically resolved propeller model applying sliding mesh interfaces is in average predicting a higher power reduction with the PSD compared to simplified propeller models. The second factor with notable influence on the power reduction prediction is the wake field prediction, which, besides numerical configuration, is affected by how hull roughness is considered.

Keywords: Ship-scale CFD, Benchmark study, Pre-Swirl Duct, KVLCC2

1. Introduction

The strive towards more fuel efficient ships is motivated by both economic and regulatory reasons. The regulatory drive stems from the target of International Maritime Organization (IMO) to reduce the total annual greenhouse gas emissions from international shipping by at least 50% by 2050 compared to 2008 [1]. A possible approach to improve the energy efficiency of a ship is to equip it with an Energy Saving Device (ESD).

The construction and function of an ESD varies widely, from those mainly aiming to reduce the hull drag to others mostly focusing on the propulsive efficiency [2]. Other approaches, such as sails or air lubrication, are also possible alternatives [3]. The ESD in focus in this study may be referred to as a pre-swirl duct (PSD), which aims to improve the propulsive efficiency through modification of the axial and tangential velocity components of the propeller inflow, at the same time as it is generating thrust and unloads the propeller. Several studies focusing on the performance of various PSDs are available in the literature [4; 5; 6; 7].

Installing a PSD is a major investment for the ship owner and prior to an order a reliable prediction of the energy savings is required. For a PSD and similar propulsion improving devices there is currently no standard for how such prediction shall be carried out and how the expected power reduction should be reported. Possible alternatives are both model-scale tests in towing tanks with associated scaling procedures, as well as different methods based on computational fluid dynamics (CFD).

Model-scale tests, and model-scale CFD, are limited by their inability to match the Reynolds number of the ship, which is practically impossible as

Froude number similarity is normally required. This implies thicker boundary layers and a larger wake on the model in relation to that on the ship. The thicker boundary layers on the model are more prone to separation. Additionally, the low Reynolds number may result in laminar boundary layers on parts of the propulsion system. A PSD is operating in the wake, and hence the differences between the model and ship wake field are critical for its performance. However, still several CFD-studies of energy saving devices are focusing on model-scale performance, as for instance [8], due to the availability of experimental data and possibility of local flow validations. Song et al. [9] shows for a specific ESD that its potential gain could be of the same order of magnitude as the uncertainties caused by scale effects, which clearly indicates its unsuitability for predictive purposes. To account for the scale-effects the International Towing Tank Conference (ITTC) has proposed specific scaling procedures. In 1999 they suggested a method where the PSD should be considered as a part of the hull in the resistance and self-propulsion tests combined with a modified scaling of the wake fraction [10]; the method was however never added to the ITTC recommended procedures and guidelines [2]. Recently, a different method has been suggested by ITTC, with another approach to obtain the ship-scale wake fraction as well as thrust deduction factor [2]. The general validity of both methods is however still an open question. Another approach is suggested for scaling of the Mewis duct, where the power saving observed for the ship is assumed to be very similar to that measured in model-scale [5]. However, this holds under the condition that the Mewis duct geometry is adjusted to the ship-scale flow based on differences between model and ship-scale CFD. One more possible method to obtain a prediction of the power reduction is to construct a wake field in model-scale that more resembles the ship wake field, as for instance conducted in [4].

An alternative to model-scale testing and associated scaling procedures, to avoid the influences from Reynolds number differences between model and ship, is the use of ship-scale CFD. The current status of ship-scale CFD for power prediction is reviewed by Terziev et al. [11], were the principal bottlenecks for replacing testing and extrapolation with ship-scale CFD are identified to be the availability of open full-scale data, including ship geometries, and computational power to predict the flow with sufficient accuracy. Model-scale CFD has reached a relatively high level of maturity through the international workshop series on CFD in Ship Hydrodynamics, held since 1980 [12]. Currently, the only to some extent corresponding workshop for ship-scale CFD is the Lloyds Register workshop held in 2016 [13]. This workshop only included overall values to use for validation, not any detailed flow measurements. However, flow measurements on ships including ship geometries, possible to use for CFD validation, are available in partly confidential data sets, as reported by for instance Inukai et al. [14], Sakamoto et al. [15], and Wakabayashi et al. [16]. There is also an industry wide research project ongoing to provide more ship-scale data possible to apply for CFD validation [17], neither this data set is presently openly available. However, the next occasion of the international workshop series on CFD in Ship Hydrodynamics [12], is planned to include a ship-scale validation case for the first time.

Despite the general lack of detailed validation data, several ship-scale

CFD studies have been conducted and published. Pereira et al. [18] conducted computations on KVLCC2 in both model and ship-scale and concluded that the scale-effects are larger than the numerical uncertainties and also that the wake-fraction reduction from model to ship-scale is clearly dependent on the selected turbulence model. Orych et al. [19] had access to both experimental data and ship trial results (confidential data) for a cargo vessel and conducted a CFD validation and verification exercise with the conclusion that there were no significant differences in uncertainty levels between model and ship-scale computations. Another study by Sun et al. [20] comparing CFD-predictions with sea trial results claimed that various free surface treatments contributed with up to 5 % uncertainty in power prediction, and that roughness could have an up to 7% effect on the delivered power. Similarly Niklas and Pruszko [21] concluded that their ship-scale CFD results varied from -10 % to +4 % in relation to sea trials data, dependent on hull roughness assumption and turbulence model. The flow measurements and calculations by Sakamoto et al. [15] indicated the necessity to account for hull roughness modelling. However, the scarce amount of data covering both flow measurements and detailed hull surface characterizations, implies that hull roughness modelling, in combination with near wall modelling and turbulence modelling, is an aspect currently associated with high levels of uncertainty.

The research area of ship-scale CFD is thus slowly evolving, but with disparate conclusions on how to prioritize the efforts and the reliability of any single computation. However, ship-owners have an urgent need for more reliable ESD energy saving predictions to be able to oblige to current and upcoming regulations. This urge motivates this study for which the objective is to compare industrial state-of-the-art ship-scale CFD predictions of the power reduction through a PSD installation on KVLCC2. The comparisons will focus on how various CFD modelling aspects influence the power reduction prediction, which hopefully can be an aid for further ship-scale CFD development and validation work, indicating where efforts on improving predictions should be focused, as well as a useful reference for ship-owners when deciding upon a possible ESD installation based on ship-scale CFD predictions. The variations in computational configurations among the benchmark submissions include, among others, choice of turbulence model, propeller modelling approach, consideration of superstructure drag, and how to include hull roughness.

2. Organization of Study

This CFD benchmark study is organized by SSPA Sweden AB and Chalmers University of Technology as part of a research project initiated due to the lack of standard for prediction of expected power reduction for ESDs and how this should be reported to a customer. All participants were invited under the condition that they had to participate at their own expenses. Everyone was supplied with the same instructions, a description of the operating conditions and ship-scale geometries, as well as CAD-files. It was clearly stated that the main aim of the study was to predict the power reduction through the PSD installation applying CFD, i.e. with less focus on the absolute power prediction. This implies that the minimum number of computations were two, self-propulsion with and without the PSD mounted on the ship, but alternative

CFD setups were warmly welcomed. The details on the CFD setup, computational grids, results and a qualitative uncertainty self-assessment were to be summarized in a provided Excel-template. The ship has never been built and hence there is no ship-scale data available for validation. Model-scale tests with and without the PSD have been conducted at SSPA, however these results have not been disclosed to any participants before the submission of the predictions. This article is accompanied by a publicly-available data set[1]. It includes instructions as provided in the CFD benchmark study, geometries and a compilation of details (CFD setup, computational grids, and results) for the submitted predictions.

3. Description of Geometry and Operating Conditions

The original KVLCC2 hull designed as a test case for CFD around 1997 is selected, which is the one used and described for instance in the 2010 Workshop in Ship Hydrodynamics [22]. Some minor geometrical modifications are introduced to the hull to obtain a watertight geometry for production purposes at SSPA, which has resulted in small differences in wetted surface area and displacement as well as LCB (longitudinal centre of buoyancy). The main particulars of the hull are provided in Table 1. It is in this benchmark study assumed that the hull is coated with a traditional anti-fouling paint which is applied according to instructions from paint manufacturers and that the measured Average Hull Roughness (AHR) can be assumed to 100 μm. Further, the transverse projected area above the water of the ship including

[1] https://figshare.com/projects/Ship-Scale_CFD_Benchmark_Study_of_a_Pre-Swirl_Duct_on_KVLCC2/133095

superstructures (A_T) is assumed to be 1200 m^2 in this study.

Table 1: Main particulars of KVLCC2.

Length between perpendiculars, L_{PP} [m]	320
Beam, B [m]	58.0
Draft, T [m]	20.8
Displacement, Δ [m^3]	312 784
Wetted surface area without rudder, S_W [m^2]	27 249
Wetted surface area of rudder, $S_W R$ [m^2]	273.3
Block coefficient, C_b	0.8098
LCB (forward of $L_{PP}/2$)	3.499%

The propeller is the one applied for the model tests with and without PSD at SSPA. It is similar to the one designed by MOERI, but not exactly the same propeller as used in previous workshops. The main particulars of the propeller are listed in Table 2. The propeller is longitudinally positioned 6.4 m from the aft perpendicular, and vertically 5.992 m above the baseline. A side view of the hull with propeller and rudder is shown in Figure 1.

Figure 1: KVLCC2 hull with propeller and rudder.

The PSD is designed by SSPA exclusively for this study. It is equipped with three stator blades as shown in Figure 2. The hull, propeller and rudder are identical for the case with the PSD mounted and for the case without

Table 2: Main particulars of propeller (R = propeller radius, P = pitch).

Propeller diameter, D_P [m]	9.86
Hub diameter, D_H [m]	1.528
Number of blades	4
Expanded blade area ratio, EAR	0.426
Chord length at 0.7R [m]	2.305
P/D_P at 0.7R	0.721
Max camber at 0.7R [m]	0.0501

PSD.

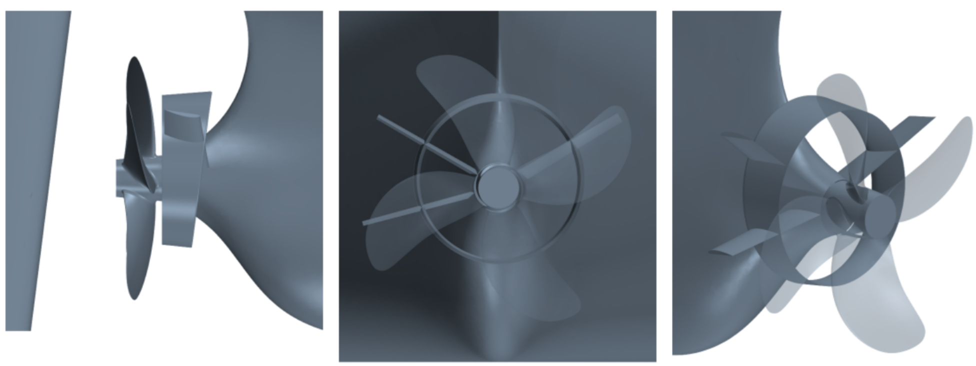

Figure 2: Pre-swirl duct (PSD) applied within this study.

The operating conditions are assumed to be optimal sea trial conditions, i.e. no currents, waves or wind to account for. The speed of the vessel applied in the CFD benchmark study is 15.0 knots, and not 15.5 knots as commonly used for KVLCC2 in workshops. 15 knots corresponds to a Reynolds number (Re) based on L_{PP} of $2.3 \cdot 10^9$ and Froude number (Fn) of 0.138. Salt water is assumed, with a water temperature of 20 °C, and air temperature 15 °C.

4. Summary of Submitted CFD-computations

In total 13 different organisations participated in the study, mainly universities and research institutes/ship-model basin organisations, but also one organization related to a software supplier and one independent CFD consultant firm. A few of them collaborated so the total number of participants should rather be counted as 10. The submitted results are obtained through the use of eight different CFD software: STAR-CCM+, FreSCo+, FINE/Marine, HELYX, OpenFOAM, NaViiX, NAGISA and SHIPFLOW. In total 22 different predictions of the power reduction through a PSD installation on KVLCC2 are done.

All predictions are based on ship-scale CFD, except from two which are model-scale CFD results extrapolated to ship-scale using scaling procedures.

4.1. CFD Setup

All submissions are based on the Reynolds-Averaged Navier-Stokes (RANS) equations. Turbulence is modelled using a variety of one- and two-equation turbulence models, namely: SST $k-\omega$, SST $k-\omega$ with QCR and curvature correction, $k-\omega$ (Wilcox), $k-\varepsilon$, Spalart-Allmaras, LEASM $k-\omega$, EASM-BSL with curvature correction, and EASM. The most frequently used model is the ordinary SST $k-\omega$ model which is applied in 12 out of 22 submissions.

The free surface is discretized using the Volume-of-fluid (VOF) method in seven submissions. For the remaining 15 submissions the setup is simplified using a symmetry plane instead of the free surface, commonly referred to as a double-body model.

Free sinkage and trim is allowed for in six of the submissions, while one submission is based on sinkage and trim results obtained in a simplified setup. The predicted sinkage and trim of the ship has a relatively low variation between submissions, and is also similar between the PSD and reference cases, for all submissions except from one outlier with significant difference in sinkage and trim between the cases. Except for the outlier, the predicted sinkage are all in the span 0.28-0.34 m and the trim predictions varies between -0.09° and -0.125° (defined as positive when bow is up).

The detailed superstructure of the ship is not known, only the assumed transverse projected area. 13 of the submissions account for air resistance using a correlation which results in air resistance of 1.3-2.4% of the total resistance of the ship, with a majority of the results within the upper range. Eight of the submissions do not account for air resistance. One submission tries to model the air resistance using a simplified superstructure, which results in a lower resistance than that obtained using the correlations.

To account for the hull surface conditions as outlined in Section 3, hull roughness is modelled in nine of the submissions using a variety of roughness functions and equivalent sand grain roughness heights. Seven of the submissions account for the additional resistance the roughness implies through a correlation, but do not model it in CFD, hence no influence on the boundary layers are accounted for. Six of the submissions do not account for the hull roughness at all.

The propeller is geometrically represented using sliding mesh interfaces in 10 of the submitted predictions. The other submissions are based on simplified propeller models. The ones applied, as described by each participant, are: a lifting line method, a hybrid lifting line/surface method (Yamazaki model), a boundary element method (BEM), a body force model combined with propeller open water curve, and a potential theory-based infinite-bladed propeller model. In one submission the propeller is geometrically represented but with the motion modelled using a moving reference frame (MRF). Amongst the submissions using sliding mesh interfaces it is most common to obtain thrust-resistance balance through manual variation of the rotation rate or to apply load variation (British method) and determine the operating point by interpolation. For the simplified propeller models it is most common with an automatic adjustment of the rotation rate.

4.2. Computational Grid

The grids are constructed using seven different software: STAR-CCM+, HEXPRESS, helyxHexMesh, snappyHexMesh, Pointwise, UP-GRID, and SHIPFLOW. The majority of the grids are unstructured, except for three submissions which apply structured grids.

All submissions, except from two, apply wall functions, where one of the submissions resolving the boundary layers is simulating the ship at model-scale. The total number of cells and average y^+ on the hull below the water surface is shown for all submissions in Figure 3. Both numbers varies widely between the submissions. No obvious correlations of the total number of cells to the free-surface modelling approach, propeller modelling approach or boundary layer resolution are noted. Amongst the submissions based on ship-scale CFD the mean number of cells is 19 million (median = 15 million) and the median value for y^+ equals 200. With regards to the boundary layer resolution, for the submissions based on ship-scale CFD, it is most common to apply a total thickness of the prism layers on the hull of 0.3-0.5% of L_{PP}, two submission apply a lower value and three a higher (for the submissions applying structured grids this factor is not relevant). The applied expansion ratio between the prism layers in the boundary layer varies between 1.2 and 1.5.

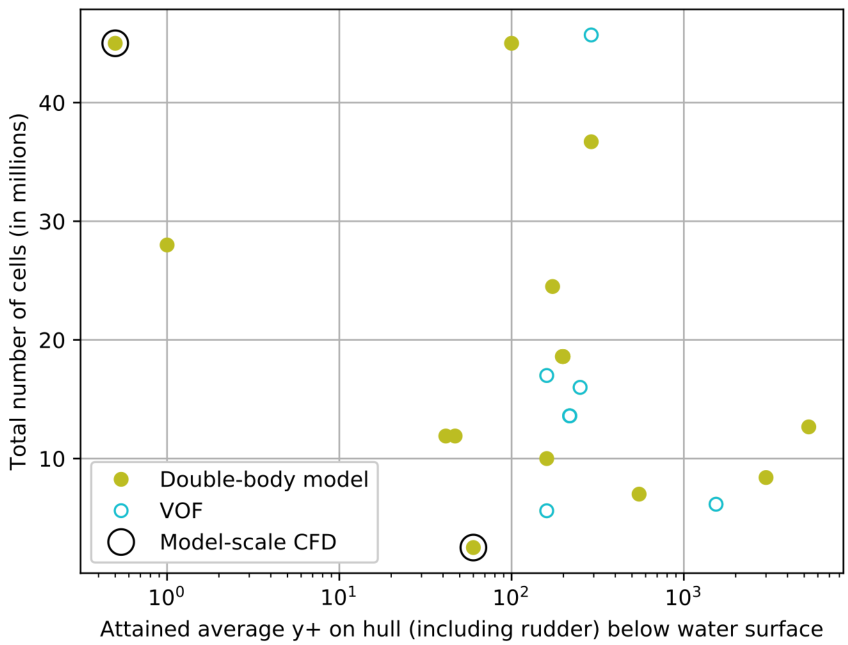

Figure 3: Total number of cells and average y^+ for all submissions.

The number of cells within the propeller domain and average y^+ on the propeller is shown for all submissions with a geometrically resolved propeller applying sliding mesh interfaces in Figure 4. A large variation is noted for both these variables amongst the submissions.

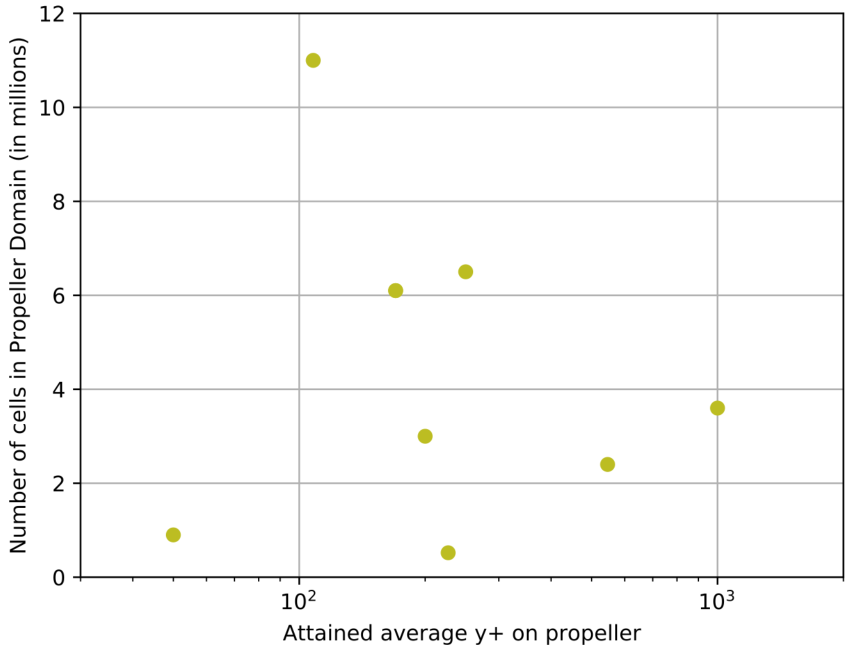

Figure 4: Total number of cells within the propeller domain and average y^+ on propeller for sliding mesh submissions.

4.3. Computational Cost

All participants have estimated the time for delivery of prognosis, counted from when the complete geometry and list of operating conditions are obtained. The estimations varies between two days and two months, but with the majority claiming approximately one week. The total required time for delivery of the prognosis is amongst other factors dependent on availability of computational resources and required number of core-hours. The total required core-hours, i.e. for both the reference case and the case with PSD, are shown in Figure 5. Except from two outliers the requirements varies between 450 and 15 400 core-hours, with a median value of 4000 core-hours. The predictions based on sliding mesh generally requires more computational resources, however there is a large variation. This variation is partially related to the required number of propeller revolutions at the final rotation rate, which varies between 5 and 600, with a median value of 24 revolutions.

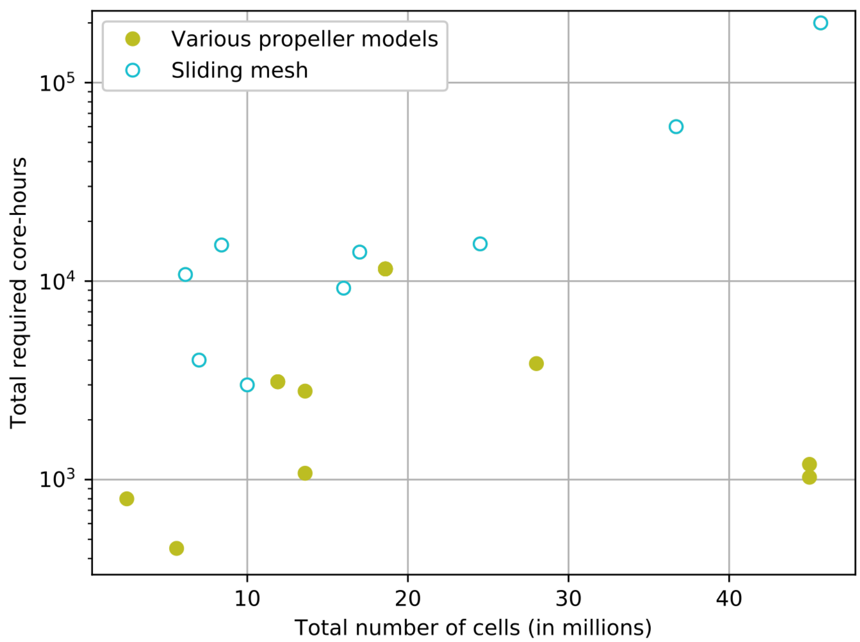

Figure 5: Total required number of core-hours in relation to the total number of cells for all submissions.

5. Comparison of Results

The main aim is to predict the power reduction through the PSD installation, hence less effort could be dedicated to the absolute power prediction.

In Figure 6 the power difference between the case with PSD and the reference case is plotted against the predicted power for the reference case. For each result it is indicated if the power prediction is considered as a representative power prediction by the user, in other words if the setup had been the same if the main aim was to predict the absolute power. The power differences are presented in relative terms, and defined so that a negative difference implies a power reduction with the PSD. Except from two outliers at about -10% and +10%, the predicted power difference through installing the PSD varies between -2.9% to +3.4%. The two outliers can be explained by large differences in sinkage and trim, as described in Section 4, and difficulties in obtaining thrust-resistance equilibrium, respectively. Due to the large uncertainties associated with these outliers, they will not be included in the further analysis of the results. Neither will the two submissions based on model-scale CFD be included in the remaining analyses, since the detailed CFD results are not comparable and additional uncertainties due to the scaling procedures are included. The average predicted power difference, with the outliers and model-scale results excluded, is -0.4% with a standard deviation of 1.6%-units.

Figure 6 illustrates also a relatively large spread in the predicted power, even when only the predictions that are considered representative by each user are taken into account. The mean predicted power amongst the predictions considered representative is 17 724 kW, with a standard deviation of 2 026 kW, corresponding to 11% in relative terms.

That the PSD is not working properly, which is indicated by the predicted power differences which varies around zero, could be partially explained by

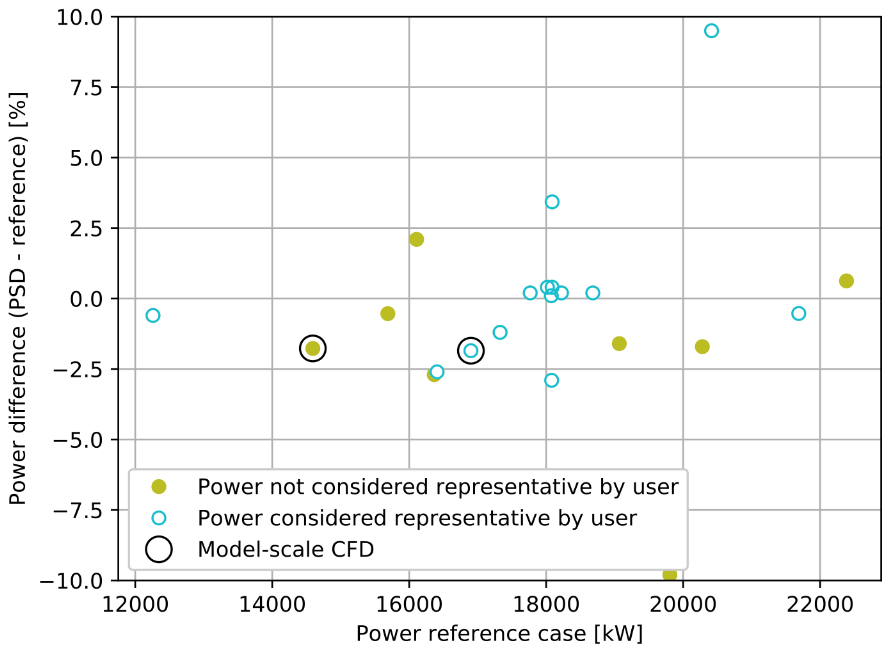

Figure 6: Power difference against absolute power for the reference case.

regions of unfavorable flow separation on the PSD indicating an unsatisfying alignment of the stators, as shown in Figure 7. Especially this is noted at the root of the stators.

Even if it is not the main objective of this study, it is interesting to compare the ship-scale CFD predictions with the model-scale test results from SSPA. Extrapolated to ship-scale, using the ITTC 1978 [23] and 1999 methods [10], the model-scale tests predict a power reduction of 4.4%, i.e. significantly more than the ship-scale CFD predictions. Further, the model-scale tests predict the power for the reference case to 16 858 kW, i.e. within one standard deviation from the mean predicted power amongst the CFD

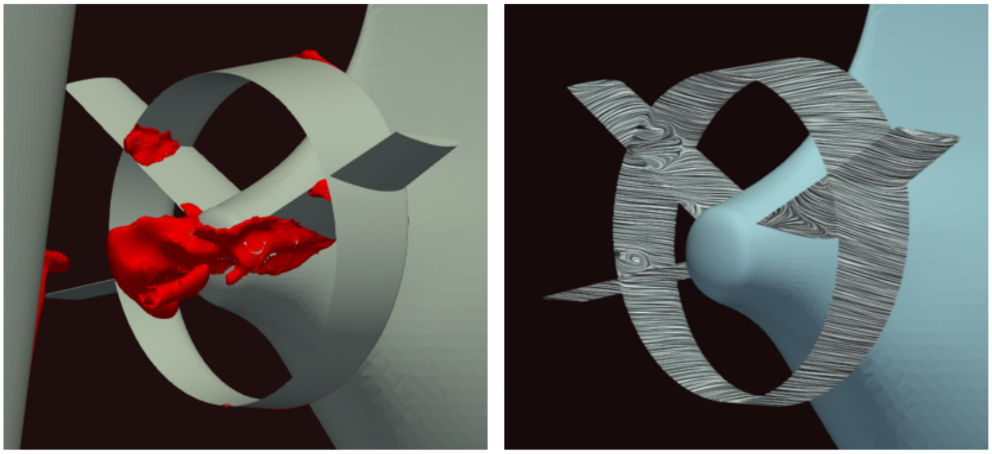

Figure 7: Iso-surface of negative axial velocity (left) and streamlines on PSD (right) illustrating the flow separation for a submission predicting a 0.2% power increment with the PSD.

submissions.

In Figure 8, the power difference is plotted against the propeller thrust difference for the data set without outliers and results based on model-scale CFD. There seems to be a correlation between the power difference and propeller thrust difference, which seems reasonable; a reduced propeller thrust implies that it is unloaded by the PSD which generates thrust. A reduced propeller thrust most probably also implies a reduced torque, which together with the rotation rate defines the power. On the other hand, when the PSD installation creates additional drag and the propeller needs to produce more thrust, an increment in power is noted if the increment in torque dominates over rotation rate differences. Additionally, Figure 9 shows the power difference against the rotation rate difference, also for the data set without outliers and predictions based on model-scale CFD.

In Figures 8 and 9 the predictions obtained using a geometrically resolved

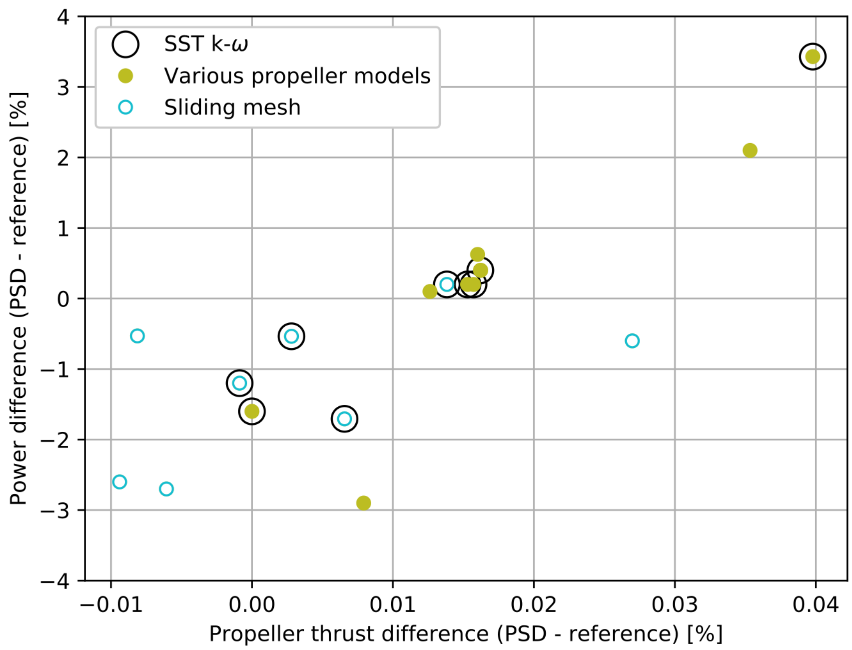

Figure 8: Power difference against propeller thrust difference.

312 propeller with sliding mesh interfaces are marked. There is an indication that the predictions applying a geometrically resolved propeller in general implies a better performance of the PSD. The average predicted power difference for the subset using sliding mesh interfaces is -1.2% with a standard deviation of 1.0%-units. For the predictions using various simplified propeller models the average predicted power difference is +0.3% with a standard deviation of 1.6%-units. Since large differences between the simplified propeller models are expected, and due to the fact that the results are influenced by other modelling aspects, a simple explanation to this observation cannot be put forward and further studies are necessary. However, a possible theory may

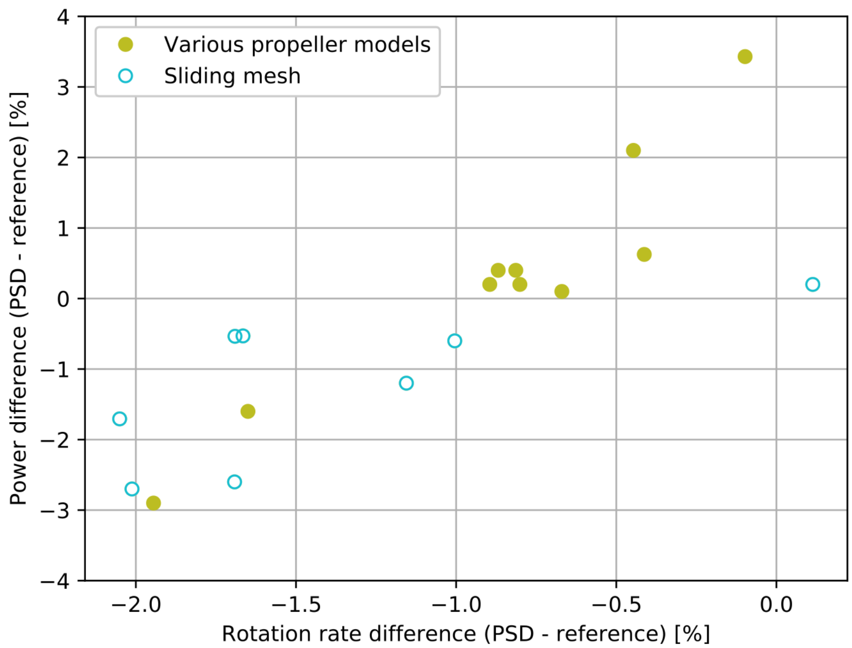

Figure 9: Power difference against rotation rate difference.

be related to the simplified propeller models representativeness at off-design conditions. Figure 10 illustrates the torque for one blade around a revolution as obtained in a submission predicting a power difference of -2.6 %. The PSD is redirecting the flow and increasing the blade torque, hence the angle of attack, especially when the blade is lightly loaded; the stators are located approximately at 60°, 255° and 300°. A simplified propeller model which is not fully representative at light load (i.e. high advance ratio), may over- or underestimate the efficiency of the propeller at these locations. If it would be so that the simplified propeller models applied in this study, to the largest extent are overestimating the propeller efficiency at light load, the propeller will not suffer as much in the reference case as a resolved propeller would indicate. Hence, the gain of adding a PSD would be lower, or even negative, as noted from the results. Additionally, simplified propeller models may lack in their ability to resolve the temporal fluctuating flow behind the stators, which may impact the possibility to accurately represent the propulsion system performance with a PSD included. It is worth to note that in a comparison in model-scale with a pre-duct without stators on the JBC test case [12], a similar trend for geometrically resolved propellers versus simplified propeller models was not seen.

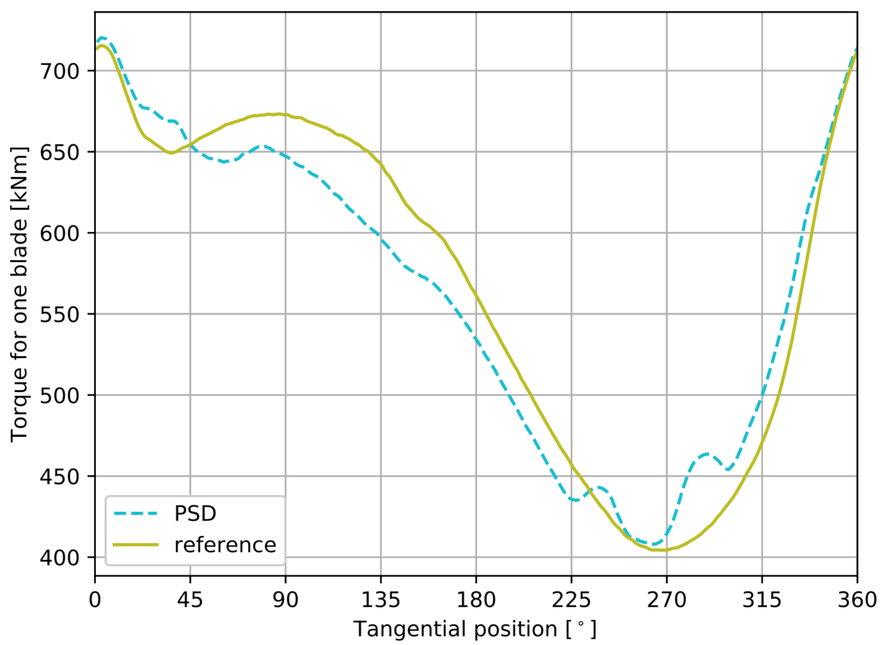

Figure 10: Torque for one blade around a revolution for one of the submissions.

Figure 8 also indicates the predictions applying the ordinary SST $k-\omega$

model, these results show a better correlation between thrust and power difference. The three main outliers in this plot are based on $k-\omega$ (Wilcox), EASM and Spalart-Allmaras. This indicates that the choice of turbulence model matters, even if there is no obvious correlation to the predicted power reduction.

The PSD operates in the wake of the ship, and hence the size and appearance of the wake field may have a strong influence on the PSD performance. A quantitative measurement of the wake is troublesome for self-propulsion cases due to the disturbance and induced flow by the propeller. To obtain a measure correlating with the size of the wake field, the fact that the wake is dependent on the thickness of the boundary layers along the hull is used. The boundary layer thickness is in turn related to the wall shear stress, which correlates with the friction velocity, u^*,

$$u^* = \sqrt{\frac{\tau_w}{\rho}} = \frac{y^+\nu}{y}, \tag{1}$$

where τ_w is the wall shear stress, ρ the density, ν is the kinematic viscosity and y the height of the near wall cell. The friction velocity naturally varies along the hull, so instead a simplified measure is applied to obtain comparable values between the submissions. The indication of the friction velocity magnitude is here based on average y$^+$ on the hull below water surface in combination with the near wall cell height at mid-ship. The predicted power difference is plotted against this variable representing the wall shear stress in Figure 11, the submission applying wall resolved grids is not included. As expected, a higher wall shear stress is generally noted for the submissions including a hull roughness model, but a large variation is noted.

One participant has investigated the influence of hull roughness through the use of the same CFD setup, applying a simplified propeller model, with and without a hull roughness model. These results are indicated in Figure 11. The difference between those two predictions is similar to the vague trend noted amongst the other predictions: an increased wall shear stress implies a better performance of the PSD. There are two outliers to this vague trend: the first one predicting a power difference of above 3% is based on a simplified propeller model; the second one does not show a benefit with the PSD installed, despite in general a very high wall shear stress. The reasons behind the second prediction is not fully understood, it is based on sliding mesh interfaces and applies the ordinary SST $k-\omega$ model. There is further scatter in the correlation between the indication of the friction velocity and the power difference, which may be attributed to for instance propeller modelling approach. However, despite this and the relatively few computations compared, the results indicate a dependency between the wake field and the PSD performance. The result is not unexpected, considering both general model-scale test experience as well as the prediction for this configuration based on operation in the larger model-scale wake of 4.4% power reduction. It highlights the importance of an accurate and relevant wake field prediction for ship-scale CFD, which puts the light on the uncertainties related to hull roughness modelling.

The dependency of the power reduction prediction on other variables, including free surface modelling approach and grid resolution does not show any trends based on this limited set of predictions with large variations in CFD setups between the participants. Further, as indicated in Figure 6 the

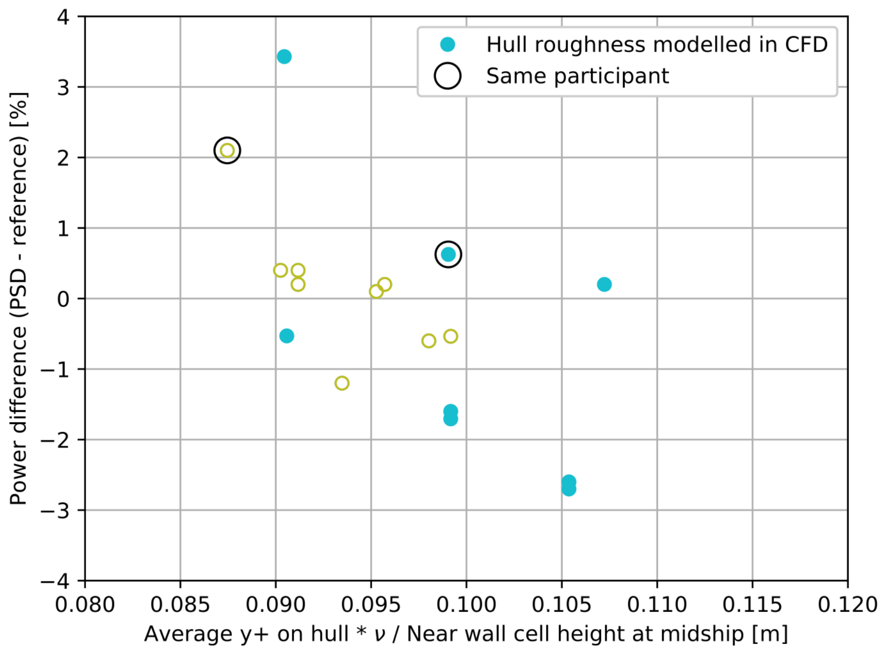

Figure 11: Power difference against an indication of the friction velocity (u^*) magnitude.

predicted power difference does not seem to have any notable correlation with the absolute power within the span predicted within this study. This implies that factors influencing the load to a minor extent, such as accounting for hull roughness only through its additional resistance or accounting for the air resistance, play a minor role.

6. Comments on Prediction Uncertainties

The predicted power gain as described in Section 5 is around zero, the average for the whole set of predictions 0.4% with a standard deviation of 1.6%-units and for the subset only including the predictions applying sliding mesh interfaces on average 1.2% with a standard deviation of 1.0%-units.

Each participant was asked to list all uncertainties related to their predictions, and assess the influence of this method/assumption on the final energy saving prediction. 10 out of the 22 submissions include such an uncertainty description. Six of the 10 estimates included uncertainties rated as "high", which was predefined as factors which may influence the power saving prediction with ±50 %, defined as that an power reduction of 1%, might as well represent a 1.5 or 0.5% power reduction. The most common uncertainties rated as "high" were propeller modelling approach and spatial discretization. Based on the variations in predicted power reduction, it also seems like the propeller modelling approach may have a large influence on the results. Interesting to note is that only four submissions brought up the hull roughness modelling as an uncertainty in their modelling at all, of which two assessed it as a moderate uncertainty and two as a low uncertainty. Considering the dependency of the power reduction against the wall shear stress, as illustrated in Figure 11, it seems like a majority of the participants underestimate the importance of the wake field prediction and hull roughness modelling. Another important remark based on the self-assessed uncertainties is that they in general are low in relation to the standard deviation of the predicted power gains.

7. Conclusions

The 10 participants, including 13 separate organizations, in this ship-scale CFD benchmark study conducted in total 22 different predictions of the power reduction through a PSD installation on KVLCC2. The predicted power reduction is varying around zero, on average 0.4%, with a standard deviation of 1.6%-units, if not considering two predictions based on model-scale CFD and two outliers associated with large uncertainties in the results. A majority of the predictions were obtained using commercially useful methods in terms of cost and delivery time, claiming approximately one week for delivery of prognosis, counted from that the complete geometry and list of operating conditions are obtained.

From this comparative study, two factors could be observed to influence the predicted power reduction: the propeller modelling approach as well as the boundary layer/wake field prediction. A geometrically resolved propeller model applying sliding mesh interfaces to simulate the propeller motion is, based on the set of submitted results, in general predicting a higher power reduction with the PSD compared to simplified propeller models. The reason behind this observation is not fully understood, but a possible theory may be that the representativeness of the simplified propeller models is lower at off-design conditions and although the propeller operates at the design point, each individual blade will experience a large variety of operating conditions during one revolution.This shows on the importance of applying the same propeller models when comparing alternatives, and also indicates that comparisons of ESD alternatives with different working principles may be sensitive to the propeller modelling approach. An indication of the boundary layer thickness is in this study obtained indirectly through a measure indicating the relative magnitude of the wall shear stress. This variable shows a vague correlation towards the predicted power reduction, where thicker boundary layers gives higher power savings. Factors influencing the wall shear stress are mainly the hull roughness modelling, but also the turbulence model and its near wall modelling.

Hull roughness modelling is a modelling aspect currently associated with high levels of uncertainty, due to scarce amount of data covering both flow measurements and detailed hull surface characterizations on ships. In this study nine out of 22 predictions account for the hull roughness in the CFD setup, using a variety of roughness functions and equivalent sand grain roughness heights, with a varying impact on the wall shear stress. This is also an aspect of highest importance for the ship-owners as it indicates that the PSD performance is dependent on the fouling rate and general hull surface condition of the ship. Further, it indicates that the PSD performance most probably varies in the period between two dry dockings, as a function of the wake field alteration.

Factors with no obvious correlation to the predicted power reduction based on the set of submitted results include free surface modelling approach, grid resolution and propeller loading (i.e. absolute power). It is however very important to keep in mind that this observation is based on a limited set of predictions with large variations in CFD setups between the participants. The absolute power seems not to correlate with the predicted power reduction. This implies that factors influencing the load negligibly, such as accounting for hull roughness only through an additional resistance or air resistance, play a minor role. The fact that several factors do not show a correlation with the predicted power reduction raises the question of the necessity to include these in the CFD-model. To exclude modelling of for instance the free-surface, super-structure drag or hull motions, is always associated with a risk since it may have an influence on the results for the specific case studied. On the other hand, all introduced modelling may also imply additional uncertainties when comparing two similar cases. This is clearly illustrated by one of the outliers in this study, where the results are heavily influenced by differences in predicted sinkage and trim.

For future studies, to increase the general maturity and trustworthiness of ship-scale CFD, the importance of flow-field measurements in combination with detailed hull surface characterizations on ships is emphasized. This could hopefully facilitate a development within the field of hull roughness modelling for ship-scale CFD which is required. The influence of the selection between alternative propeller models still needs further work, particularly with their applicability to a wide range of operational conditions. While it is attractive, especially for design optimization to use a lower computational cost approach, this cannot be at the expense of failing to resolve the physics of the performance gain associated with an ESD.

Acknowledgements

The organization of the study and the contributions by Chalmers and SSPA was supported by the Swedish Transport Administration (grant number TRV 2018/76544).

The contribution by Chalmers was based on simulations performed on resources at the National Supercomputer Centre in Sweden (NSC), provided by the Swedish National Infrastructure for Computing (SNIC).

The contribution by University of Strathclyde's were based on results obtained using the Archie-West High Performance Computer (www.archie-west.ac.uk).

The contribution by ECN/CNRS was granted access to the HPC resources of IDRIS under the allocation 2021-A0092A01308 made by GENCI.

References

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

[2] I. Lee, J. Gose, A. Coraddu, J. Chen, M. Hinatsu, R. Quereda, T. Li, The Specialist Committee on Energy Saving Methods: Final Report and Recommendations to the 29th ITTC (2021).

[3] A. F. Molland, S. R. Turnock, D. A. Hudson, I. K. A. P. Utama, Reducing ship emissions: a review of potential practical improvements in the propulsive efficiency of future ships, International Journal of Maritime Engineering 156 (2014) 175–188.

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

[5] T. Guiard, S. Leonard, F. Mewis, The Becker Mewis Duct - Challenges in Full-Scale Design and new Developments for Fast Ships, in: Pro-

ceedings of the Third International Symposium on Marine Propulsors, Launceston, Tasmania, Austrailia, 2013.

[6] J. H. Kim, J. E. Choi, B. J. Choi, S. H. Chung, H. W. Seo, Development of Energy-Saving devices for a full Slow-Speed ship through improving propulsion performance, International Journal of Naval Architecture and Ocean Engineering 7 (2) (2015) 390–398.

[7] H. Nowruzi, A. Najafi, An experimental and CFD study on the effects of different pre-swirl ducts on propulsion performance of series 60 ship, Ocean Engineering 173 (2019) 491–509.

[8] N. Sakamoto, K. Kume, Y. Kawanami, H. Kamiirisa, K. Mokuo, M. Tamashima, Evaluation of hydrodynamic performance of pre-swirl and post-swirl ESDs for merchant ships by numerical towing tank procedure, Ocean Engineering 178 (2019) 104–133.

[9] K. w. Song, C. y. Guo, C. Wang, C. Sun, P. Li, R. f. Zhong, Experimental and numerical study on the scale effect of stern flap on ship resistance and flow field, Ships and Offshore Structures 15 (9) (2020) 981–997.

[10] ITTC, The Specialist Committee on Unconventional Propulsors: Final Report and Recommendations to the 22nd ITTC (1999).

[11] M. Terziev, T. Tezdogan, A. Incecik, Scale effects and full-scale ship hydrodynamics: A review, Ocean Engineering 245.

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

[13] D. Ponkratov, Proceedings of 2016 Workshop on Ship Scale Hydrodynamic Computer Simulation (2017).

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

[15] N. Sakamoto, H. Kobayashi, K. Ohashi, Y. Kawanami, B. Windén, 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.

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

[17] JORES, Development of an industry recognised benchmark for Ship Energy Efficiency Solutions (2019).
URL `https://jores.net/`

[18] F. S. Pereira, L. Eça, G. Vaz, Verification and Validation exercises for the flow around the KVLCC2 tanker at model and full-scale Reynolds numbers, Ocean Engineering 129 (2017) 133–148.

[19] M. Orych, S. Werner, L. Larsson, Validation of full-scale delivered power CFD simulations, Ocean Engineering 238.

[20] W. Sun, Q. Hu, S. Hu, J. Su, J. Xu, J. Wei, G. Huang, Numerical analysis of full-scale ship self-propulsion performance with direct comparison to statistical sea trail results, Journal of Marine Science and Engineering 8 (1).

[21] K. Niklas, H. Pruszko, Full-scale CFD simulations for the determination of ship resistance as a rational, alternative method to towing tank experiments, Ocean Engineering 190.

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

[23] ITTC, 1978 ITTC Performance Prediction Method. Recommended Procedure 7.5 - 02 - 03 - 01.4 Rev 04 (2017).