Propulsion Committee

Final Report and Recommendations to the 27th ITTC

1 INTRODUCTION

1.1 Membership and Meetings

The members of the Propulsion Committee of the 27th International Towing Tank Conference are as follows:

- Dr. Didier FRECHOU (Chairman), DGA Hydrodynamics, France
- Tom DINHAM-PEREN (Secretary), BMT Defence Services Ltd, U.K.
- Rainer GRABERT, Schiffbau-Versuchsanstalt Potsdam GmbH (SVA) Germany
- Valery BORUSEVICH, Krylov State Research Center, Russia
- Professor Chen-Jun YANG, Shanghai Jiao Tong University, China
- Professor Emin KORKUT, Istanbul Technical University, Turkey
- Professor Steven CECCIO, University of Michigan, USA
- Takuya OHMORI, Japan Marine United Corporation, Japan
- Professor Moon Chan KIM. Pusan University, Korea

Five Committee meetings were held as follows:

- DGA Hydrodynamics, France, 30-31 January 2012
- Krylov Institute, Russia, 8-9-10 October 2012
- Pusan University, Korea, 22nd and 23rd January 2013
- University of Michigan, USA, 23-25 October, 2013.
- BMT Defence Services Ltd, UK, 13-14 March 2014.

1.2 Recommendations of the 26th ITTC

The 26th ITTC recommended the following tasks for the 27th ITTC Propulsion Committee:

1. Provide an update of the state-of-the-art for predicting propulsion systems emphasing developments since the 2011 ITTC Conference. The committee report includes discussions of the following topics:

   a. The potential impact of new technological developments on the ITTC, including new types of propulsors, azimuthing thrusters, and propulsors with flexible blades.

b. New experimental techniques and extrapolation methods.
c. New benchmark data.
d. The practical applications of computational methods to the propulsion systems predictions and scaling.
e. New developments of experimental and CFD methods applicable to the prediction of cavitation.
f. The need for R&D for improving methods of model experiments, numerical modeling and full-scale measurements.
g. A review of new developments regarding high-speed marine vehicles

2. Provide a review of ITTC Recommended Procedures relevant to propulsion. The committee report specifically discusses the following topics:

   a. Identification of needed changes in procedures the light of current practice, and, if approved by the Advisory Council, provision of updated requirements.
   b. Identification of any needed new procedures, including an outline of their purpose and content.

3. Liaise with the Specialist Committee on Performance of Ships in Service, especially regarding power prediction and consequences of EEDI.

4. Assess where CFD results can be introduced to support experimental model testing by monitoring status of CFD to perform full scale powering, resistance, cavitation and wake simulations and their correlation with full scale data. Identify the needs for hybrid procedures combining experimental and numerical methods.

5. Prepare a state-of-the-art review of modelling and scaling unconventional propulsion and wake improving devices.

6. Examine methods of target wake simulation, e.g. the "smart dummy" approach.

7. Examine wake fraction scaling for twin-screw ships, and show the consequences on existing procedures.

8. Examine the possibilities of CFD methods regarding scaling of conventional and unconventional propeller open water data, including initiation of a comparative CFD-calculation project.

9. Develop guidelines for hybrid propulsor testing.

10. Continue monitoring existing full-scale data for podded propulsion, if such data is available.

1.3 General Remarks

All the tasks outlined in the terms of reference were taken in charge by the present committee. The committee had some difficulties liaising with other committees concerning Task 3 and Task 4. The portion of this report regarding procedural reviews has been recently reported to the AC, which recommended that the procedures be a continuing consideration of the next committee. Concerning the CFD comparative benchmark, a joint effort of the Propulsion committee and the CFD committee will continue to gather contributions from all the ITTC organisations.

2 STATE OF THE ART UPDATE

Many major international conferences were held since the 26th ITTC conference in 2011:

- 9th Symposium on Particle Image Velocimetry, 21-23 July 2011, Kobe.
- ICOMIA's 1st International Hybrid Marine Propulsion Conference, November 2011, The RAI, Amsterdam.
- SMP11 International Symposium on Marine Propulsors and Workshop, June 2011, Hamburg.
- IWSH 2011: 7th International Workshop on Ship Hydrodynamics 16-19 September 2011, Shanghai.
- MARINE 2011- IV International Conference on computational methods in marine Engineering, 28-30 September, Lisbon.
- IMDC 2012-11th International Marine Design Conference, June 2012, Glasgow.
- ICHD 2012- The 10th International Conference on Hydrodynamics, 1 - 4 October, 2012, St Petersburg.
- Voith Hydrodynamic conference, June 2012.
- CAV2012- 8th Symposium on Cavitation, 14-16 August, 2012, Singapore.
- ICMT 2012- International Conference on Maritime Technology, 25-28 June 2012, Harbin.
- ONR 29th symposium on Naval Hydrodynamics, 24 August 2012, Goteborg.
- Journées de l'Hydrodynamique 2012, 21-22-23 Nov 2012, Paris.
- NAV'2012 - 17th International conference on Ships and Shipping Research 17-19 October 2012, Naples.
- ICETECH 2012, International Conference and Exhibition on Performance of Ships and Structures in Ice, September 17-20, 2012, Banff,
- 13th Propeller/Shafting Symposium September 11 – 12, 2012, Norfolk.
- ONR Naval S&T Partnership Conference event, October 22-24, 2012, Washington D. C.
- IWSH'2011, The 7th International Workshop on Ship Hydrodynamics, 16-19 September, 2011, Shanghai.
- ISOPE 2012 Conference: 22nd international Ocean and Polar Eng, 17-23 June, Rhodes.
- HIPER, 28-29 Sept 2012, Duisburg.
- ISOPE 2013 Anchorage Conference: 22nd international Ocean and Polar Eng, 30 June – 4 July, Anchorage.
- PRADS 2013: The 12th International Symposium on Practical Design of Ships and Other Floating Structures, 20-25 October 2013, Changwon.
- FAST 2013, 12th International Conference on Fast Sea Transportation, 2-5 Dec 2013, Amsterdam.
- AMT 2013, The 3rd International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, 17-19 September 2013, Gdansk.
- OMAE 2013, The 32nd International Conference on Ocean, Offshore and Arctic Engineering, June 9 to 14, 2013, Nantes.
- IWSH 2013: The 8th International Workshop on Ship Hydrodynamics, 23- 25 September, 2013, Seoul.
- SMP'13, The Third International Symposium on Marine Propulsors, 5 – 8 May, 2013, Launceston.

The most relevant papers from these conferences and from other technical journals and conferences were reviewed and reported.

2.1 New technological developments

2.1.1 New types of propulsors

There is still a tremendous interest concerning Contra-Rotating Propeller (CRP) concepts based on combination of conventional propellers and Pods, either on single or twin shafts (see examples on figures 1, 2, 3 & 4)

Figure 1: The first ferry with a podded CRP propulsion system (Ueda *et al.*, 2004)

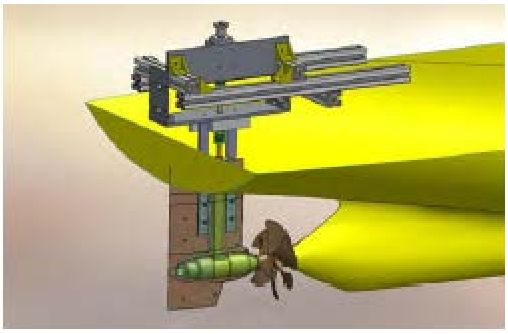


Figure 2: CRP Combination of a Rudder Pod unit and a single propeller (Sánchez-Caja, *et al.*, 2013)

Figure 3: The main propeller (right) with counter rotating, 360 degree azimuthing, ABB Azipod thruster on 200 TEU Container Feeder vessel (Henderson, 2013)

One advantage of a CRP system when compared to a single propeller is that, as the two propellers of the CRP share the total propulsive force, the load on a single propeller is reduced, allowing for a reduction in rotation speed. Thus, increased propulsion efficiency can be obtained compared to a single propeller of the same diameter.

Figure 4: CRP Electric Propulsion system (Hideki, *et al.* 2011)

As pointed out by Hideki *et al.* (2011), in an electric propulsion vessel, there is no need to connect a large main engine directly to the propeller shaft. Instead, two electric propulsion motors much smaller in size than the main engine are connected to the propeller shaft through a CR gear. Since the electric propulsion devices are connected *via* electrical buses, the arrangement in the engine room is more flexible than in conventional vessels (Figure 5).

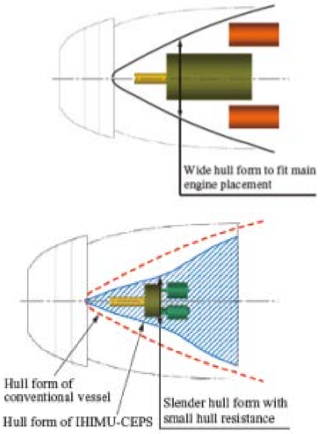


Figure 5: Flexible engine room arrangement and development of hull form (Hideki *et al.*, 2011)

Therefore, the hull form from midships to the stern, which is important to reducing fluid resistance, is can be improved compared to vessels with conventional propulsion (Fig. 6).

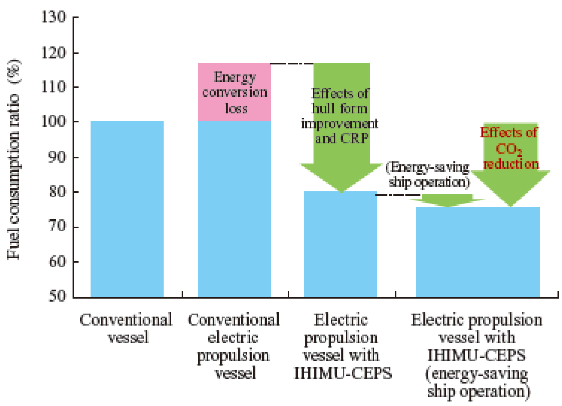


Figure 6: Comparison of fuel consumption between conventional vessel and electric propulsion vessel with IHIMU-CEPS (1,230 m^3 type chemical tanker used as an example) (Hideki *et al.*, 2011)

Most cargo vessels have only a single propeller directly driven by a diesel engine. Single screw propulsion offers an efficient hull form. The resistance is lower and the hull efficiency is high owing to the beneficial wake field behind the skeg. This configuration provides the most cost efficient solution for most cargo vessels with modest power, large drafts and little demands on manoeuvring performance and redundancy.

Full displacement ferries, on the other hand, usually have twin screws and multi-engine machinery. There have of course been good reasons for these trends. Ferries are often faster and require increased propulsive power. Their draft is often limited, and the propeller loads becomes higher. These factors favour twin-screw solutions, where the power can be divided between two propellers. Safety aspects and fast turnaround in port favours two propellers.

A range of new propulsion concepts for ferries have been presented in recent years, such as Podded CRP, Wing Pods and Wing Thrusters. These have some features in common in that they do not use a traditional twin shaft line arrangement but instead employ a propeller mounted on the centreline skeg combined with either one or two azimuthing propulsors.

Several recent papers reveal an increasing interest in Energy saving devices before or after the propeller or within the propeller itself. The review of new developments on that topic is largely detailed in Section 6.

A few projects using immersed pump-jet or water-jet have also been published. Pospiech (2012) presented a design of a pump-jet fully integrated with the ship hull (Figure 7). Giles *et al.* (2011) presented a design of water-jet fully immersed and also fully integrated within the ship hull (Figure 8).

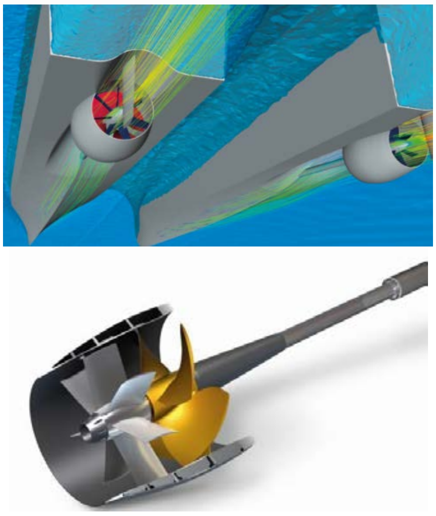


Figure 7: VOITH's New Propulsion System: The Voith Linear Jet (Pospiech, 2012)

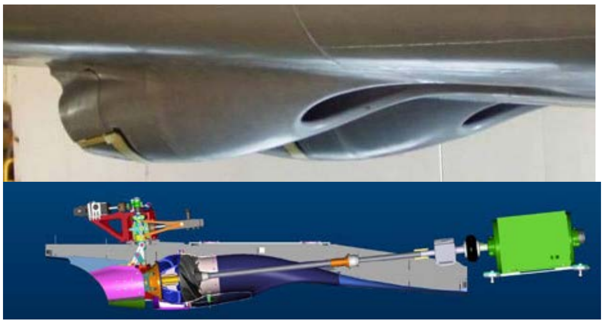


Figure 8 : WaterJet: Propulsor (Giles, *et al.* 2011)

Although the Hybrid sailing vessel is still at a research and development stage, expected fuel energy savings are very promising. Using 9 rigid sails on a cap-size bulker of 180,000DW, Ouchi *et al.* (2013) forecast a fuel energy savings of at least 20%. CFD was used to estimate the thrust distribution on every sail for different apparent wind angles.

Figure 9: Hybrid sailing vessel (Ouchi, *et al.* 2013)

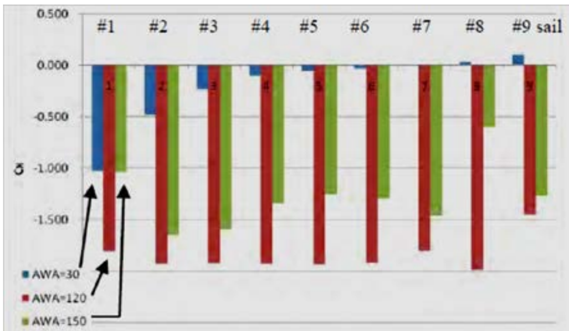


Figure 10: Thrust force distribution on sails (Ouchi, *et al.* 2013)

The potential impact of these new propulsors are listed below:

- New procedures are required for self-propulsion test of contra-rotating propulsion system and for pump-jets that are integrated within the hull
- CFD calculation might be required to support EFD to assess the performance of Energy Saving Devices.
- A new procedure for self-propulsion will certainly be required for hybrid sailing vessels.

2.1.2 Azimuthing thrusters

Two papers (Palm, *et al.*, 2011; Koushan, *et al.*, 2011) have shown interests on the effect of ventilation on azimuthing thruster performances.

Palm *et al.* (2011) present a comparative study between cycloidal propeller and azimuthing thruster, investigating the effect ventilation on the thrust losses. The blade thrust force of an azimuthing thruster is subject to large variations when ventilation is occurring. Due to its working principle, the cycloidal propeller is less prone to ventilation than the azimuthing thruster.

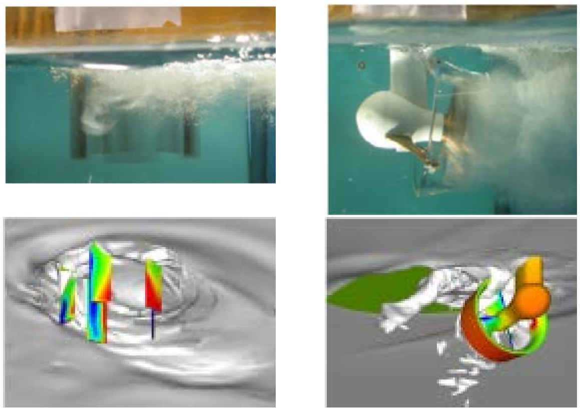


Figure 11: Cycloidal propeller and azimuthing thruster at ventilation conditions (Palm *et al.*, 2011)

Koushan *et al.*, 2011 present a similar study on ventilated propeller blade loadings and spindle moment of a thruster in calm water and waves. Experimental results are presented for the thrust, torque, and spindle moment of a single blade of a propeller from a pulling thruster under various ventilated operating conditions and in waves.

Figure 12: Azipull Thruster (Koushan *et al.*, 2011)

For all ventilated conditions, it can be observed that a sudden drop in thrust is measured when the advance coefficient becomes less than a critical advance coefficient, which is $J \approx 0.6$ for calm water and $J \approx 0.5$ for wave conditions. From the critical advance coefficient down to bollard condition, further reduction of the thrust is occurs, though slight thrust recovery is registered close to the bollard condition ($J = 0$). Dynamic variations are analysed using standard deviation and histograms. As histograms approximate the probability distribution, they show that the standard deviation values should be handled with some care as the data shows distributions that can be both highly skewed and non-Gaussian. The effect of waves and ventilation on propeller torque follows the same trends as on propeller thrust. It is observed that the spindle moment changes sign from positive to negative at high J values.

Amini & Steen (2011) performed a series of model tests on an azimuth thruster model in oblique inflow conditions for different heading angles and at different advance coefficients in pushing and pulling modes. Tests were performed in ventilating and non-ventilating conditions. A novel shaft dynamometer was used to measure all six component forces and moments on the propeller shaft. It was found that the propeller shaft lateral force and bending moment were quite large, and thus, the load at the shaft-bearing positions was about three times larger than when only the propeller weight was considered. The results also showed that oblique inflow due to steering gives higher bending loads than when the propeller is subject to ventilation in the straight-ahead condition. A basic blade element momentum method (BEMT) was used to predict the forces and moments on the propeller shaft in oblique flow conditions. Fairly good agreement was found between the BEMT results and the experimental results.

The authors finally recommend considering the shaft side forces and bending moments due to steering and oblique inflow in the mechanical design of the propeller suspension such as thruster housing and propeller shaft bearings.

2.1.3 Flexible blade propulsors

Composite marine structures are attractive because of their ability to conserve weight, reduce maintenance cost, and improve performance via 3-D passive hydroelastic tailoring of the load-dependent deformations. Manufacturers are proposing carbon fibre propeller of diameter up to 3m.

Several attempts have been made to manufacture full size composite propellers and some trials have been conducted in the recent past to compare composite structure and Nickel Aluminium Bronze (NAB) casting.

In 2000, QinetiQ investigated a 2.9 m diameter composite propeller on the Research Vessel Triton, a triple hull warship, nowadays used as a patrol vessel by the Australian Customs. This composite propeller consists of five composite blades bolted and bonded to a NAB hub. As mentioned by Kane (2001), it was designed to explore the mechanical properties required in this application include mechanical performance (stiffness, strength and fatigue) as well cavitation inception speed, reported to be 30% higher than the original NAB propeller.

In another example, Airborne Composites successfully developed composite propeller blades for the Royal Dutch Navy, supplying them with a composite main propeller for an Alkmaar-class mine hunter (Figure 13). This propeller is for a power of about 1400 kW and has a diameter of 2.5m (Black, 2011).

Figure 13: Composite propeller to the RNLN minehunter. (Black, 2011)

Few experiments have been performed at model scale, one example is Taketani *et al.*, 2013 where different propeller materials have been tested (Figure 14). The results show that the propeller “C” (sintered nylon powders with a laser heating source) presents larger blade deviations than carbon composite material. For a same propeller loading (K_t), the advance ratio J is significantly reduced (Figure 15).

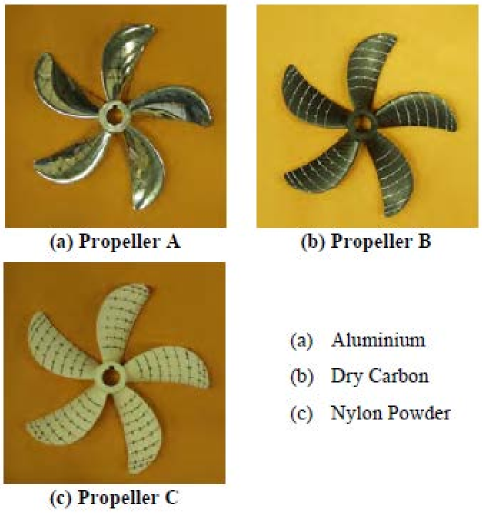


Figure 14: Model composite propeller (Taketani *et al.*, 2013)

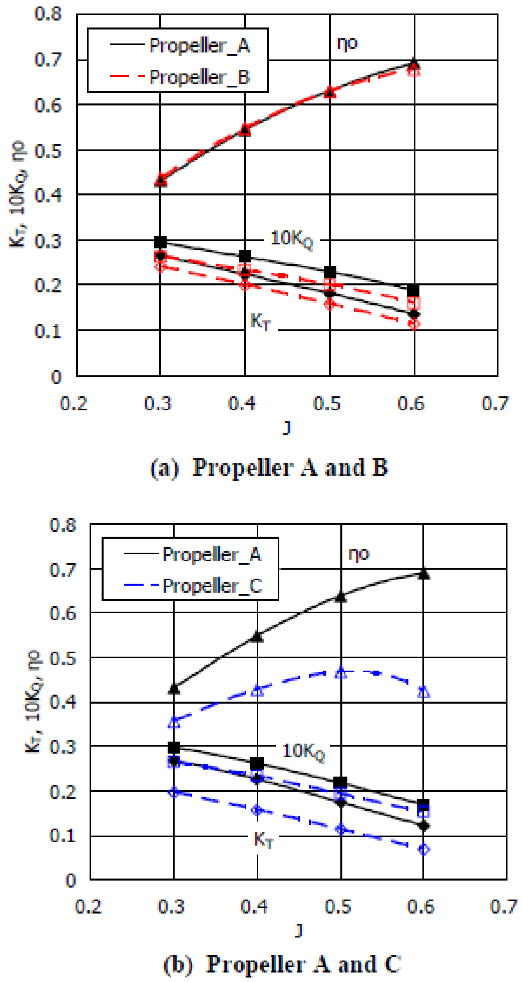


Figure 15: Model composite propeller performances (Taketani *et al.*, 2013)

Propeller efficiency increased in cases where this deformation was small (Dry Carbon), since the loss of torque was greater than the loss in thrust. At a certain point, propeller efficiency begins to decline with greater deformation, suggesting an optimal level of deformation. Elastic deformation was dominant at the blade tip. This deformation occurred along the direction of thrust and worked as a forward rake.

The cavitation generated after deformation indicated that such deformation reduced loads at the blade tip and affected pitch angles. This deformation is expected a reduction of pressure fluctuations.

In Fluid-Structure Interaction (FSI) analysis, calculations results and model test results were compared. In the case of small deformations, analysis results were consistent with changes in propeller characteristics. For larger deformations, analysis proved relatively inaccurate in estimating the deformation of the entire propeller. Future efforts should target improvements in this aspect.

Manudha, *et al.* (2013) presented a validation study that compares results obtained numerically using Fluid-Structure Interaction of Finite Element Analysis and experimental results. This validation has been carried on a twisted-bend-twisted coupled hydrofoil. Although several simplifications were made for modelling purposes, the consistency between Finite Element Analysis and experimental results were found in good agreement. The knowledge gained through this validation study is extremely helpful in developing an optimisation scheme and an accompanying numerical model that can accurately predict the performance of optimised designs without the need for extensive experimentations.

Extensive studies have been made by Young (Young, 2007; Young, 2010; Young, 2012; Motley & Young 2012) on flexible blade propellers. Among all those studies, the impact on similarity to be applied for model tests on flexible blade propellers (Figure 16) in order to scale the fluid structure response is of major interest.

Figure 16: Elastic blade deformation on a composite propeller (Young, 2012).

Young (2010) presents a detailed analysis of the dynamic hydroelastic scaling of self-adaptive composite marine rotors. The scaling analysis main goal is to define how to achieve the same dynamic load-deformation responses between the model and the prototype. This can be achieved by requiring the model to be geometrically similar to the prototype, by requiring the effective structural mass and structural rigidities to be the same and by requiring the same flow velocity as at full scale: $V_M = V_{FS}$

The following is a simpler way of presenting the implication of the similarity laws. When we have to consider testing at model scale with flexible blades, the strain should be kept the same between model scale and full scale. This is to ensure that the displacement, induced by the elasticity of the blade and which changes the angle of attack, will be scaled between model and full scale (Figure 17).

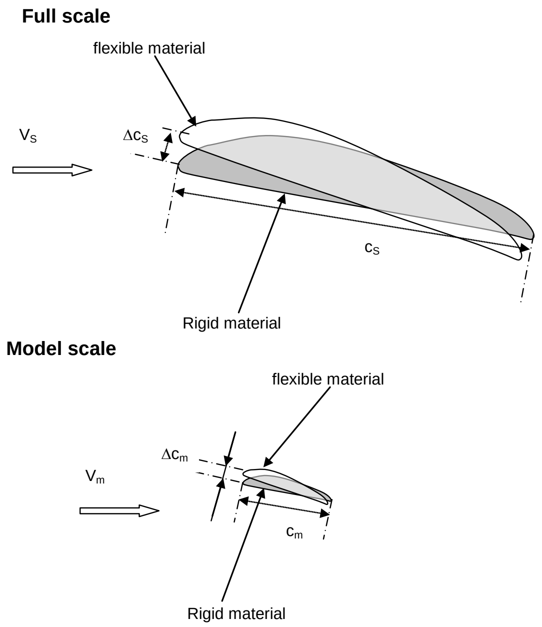


Figure 17: Strain at full scale and model scale.

This is equivalent to say that the strain on the blade section, that could be defined as the ratio $\varepsilon = \dfrac{\Delta c}{c}$ as shown on the figure, should be the same at model scale and full scale. The strain is related to the stress σ by the modulus of elasticity E:

$$\varepsilon = \frac{\Delta c}{c} = \frac{\sigma}{E}$$

c : chord length
Δc : displacement
E : is the modulus of elasticity
σ : is the stress in the material

On the other hand, the stress is a function of hydrodynamic forces and centrifugal forces which means that we can write:

$$\sigma = f\left(F_{Hydro}, F_{Centrifugal}\right)$$

$$F_{Hydro} \leftrightarrow \frac{1}{2}\rho \cdot V^2 \cdot S \Leftrightarrow F_{Hydro} \leftrightarrow \rho \cdot V^2 \cdot D^2$$

$$F_{Centrifugal} \leftrightarrow m_{prop} \cdot \gamma = m_{prop} \cdot \omega^2 \cdot R$$

$$\Leftrightarrow F_{Centrifugal} \leftrightarrow \rho_{prop} \cdot D^3 \cdot \omega^2 \cdot \frac{D}{2}$$

$$\Leftrightarrow F_{Centrifugal} \leftrightarrow \rho_{prop} \cdot V^2 \cdot D^2$$

To be more accurate, added mass in addition to the mass of the blade should be taken into account. But the blade mass as well as the fluid added mass can both be scaled by a factor.

The similarity between model scale and full scale implies that the same ratio of hydrodynamic force to centrifugal forces should be kept the same at model and full scale which means the ratio of water density to propeller density should be kept the same:

$$\frac{\rho \cdot V^2 \cdot D^2}{\rho_{prop} \cdot V^2 \cdot D^2} = \frac{\rho}{\rho_{prop}}$$

Because stress is homogeneous to a pressure, we can write that the stress between model scale and full scale is:

$$\frac{\sigma_M}{\rho_M \cdot V_M^2} = \frac{\sigma_S}{\rho_S \cdot V_S^2}$$

In order to get the same kind of strain on the blades as at full scale, this leads to the following relationship :

$$\varepsilon_S = \varepsilon_m \Leftrightarrow \frac{\sigma_S}{E_S} = \frac{\sigma_m}{E_m}$$

Combining all those similarity rules, we find that:

$$\left.\begin{aligned} &\varepsilon_S = \varepsilon_M \Leftrightarrow \frac{\sigma_S}{E_S} = \frac{\sigma_M}{E_M} \\ &\frac{\sigma_M}{\rho_M \cdot V_M^2} = \frac{\sigma_S}{\rho_S \cdot V_S^2} \\ &\left.\frac{\rho}{\rho_{prop}}\right|_M = \left.\frac{\rho}{\rho_{prop}}\right|_S \end{aligned}\right| \Rightarrow \begin{aligned} \rho_{prop\,M} &= \rho_{prop\,S} \\ E_M &= E_S \\ V_M &= V_S \end{aligned}$$

This demonstrates that with same blade material at full scale and with the same full scale speed, the strains and loading will be the same.

However, it is problematic to run model test following these requirements, as the similarity conditions are very difficult to achieve on the model scale for the followings reasons:

1. Manufacture of the model scale propeller (*e.g.* 250 mm in diameter) is required to have the same isotropic material properties as that of the full scale (*e.g.* 2.5 m) propeller; this is difficult to achieve, especially with composite carbon fibre materials at the root and the tip.
2. Testing at full-scale speeds necessitates performing the test at very high static pressures in order to avoid any cavitation. In practice, those conditions cannot be achieved in a towing tank, but they can be achieved in cavitation tunnel.

It seems more reasonable to use CFD which might use a combination of fluid and structural modelling (Young, 2007; He *et al.*, 2012). Meanwhile special care should be taken for the composite structural characteristics, for, as pointed out by Young & Motley (2011), the variations in material parameters and material failure initiation models lead to a much wider spread of propeller performance characteristics, operating conditions, and safe operating envelopes for an adaptive propeller compared to a rigid propeller.

2.1.4 Podded propeller in Ice and bubbly flow

Due to the growing interest of a potential new northern route induced by the global warming, several studies have been carried on propeller ice blade load impact (Brouwer *et al.*, 2013; Sampson *et al.*, 2013), propeller wash (Ferrieri *et al.*, 2013), and cavitation (Sampson & Atlar, 2013).

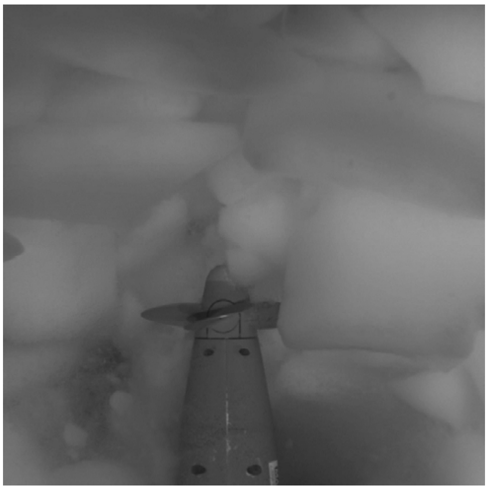


Figure 18: Picture from a high-speed camera of a propeller entering an ice ridge (Brouwer *et al.*, 2013).

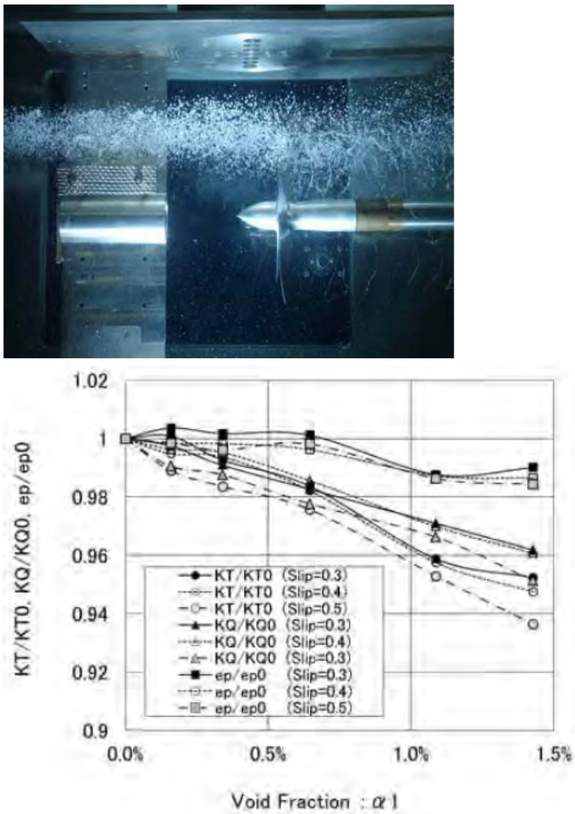


Figure 19: Propeller Thrust, Torque reduction as a function of void fraction (Kawakita, 2013)

Brouwer *et al.* (2013) developed a measurement setup in which a 6 components balance is measuring the 6 degrees of freedom forces and moments on a single blade of the podded propeller. This measurement setup was used to measure the ice impact in a model test at the AARC facility of Helsinki.

A few papers (Kawakita *et al.* 2011, and Kawakita, 2013) have discussed the effect of air lubrication of a ship hull on propeller efficiency. In recent years, air lubrication systems have been attracting attention as a method of reducing carbon dioxide emissions from ships by reducing the total resistance of the ship. The bubbly flow generated travels to the stern such that the propeller may partially work in two-phase flow modifying the thrust and torque (Figure 19).

2.2 New Experimental Techniques and Extrapolation Methods

2.2.1 3D flow visualization

Only one paper dealing with 3D flow investigations around the propeller caught the attention of the committee. Pecoraro *et al.* (2013), present a 3D flow velocity measurements, using the LDV technique. The major outcome of the analysis is that the effect of the propeller suction, which increases the velocity, extends upstream at a distance about 1 propeller radius, and that the flow fluctuation induced by the blade passage extends upstream to a distance of about 4 propeller radii. The propeller is able to reduce the size of the detached area longitudinally and transversally but is not able to remove totally the flow separation. The boundary of the separated flow can be identified by using the skewness coefficient, which allows a better identification of the extension of the separated flow.

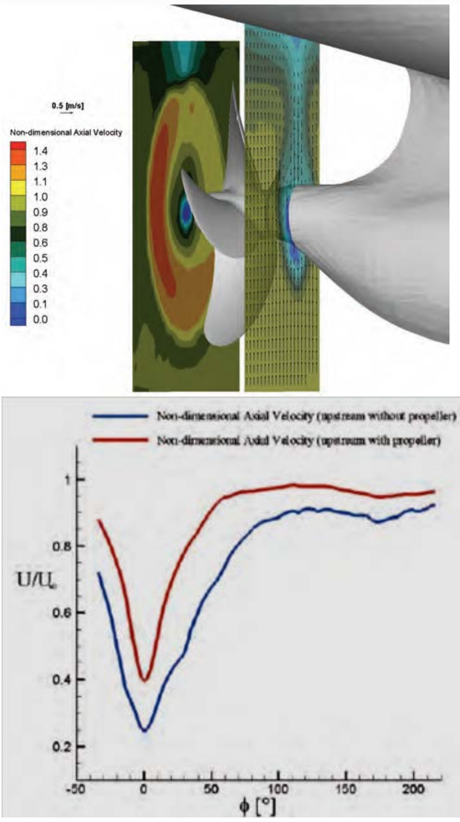


Figure 20: Axial velocity at R/R=0.7 with and without the propeller (Pecoraro *et al.*, 2013)

Manufacture / control of Model Propellers

For measurement of blade geometry, Dang *et al.* (2012) are presenting a new optical technique based on digital photogrammetry. In order to have control on the accuracy of the blades, the propeller is optically scanned at its design pitch. The results are compared to the theoretical geometry and the deviations are determined and presented in 3D images of the model propeller (Figure 21).

2.2.2 Propeller manufacturing

Only one paper concerning new techniques for manufacturing model propeller, (Taketani *et al.*, 2013) was found. A model propeller was manufactured by sintering nylon powders with a laser and was then compared with aluminum made propeller and carbon composite propeller (see Figure 14). These results of this work are discussed in Section 2.1.3.

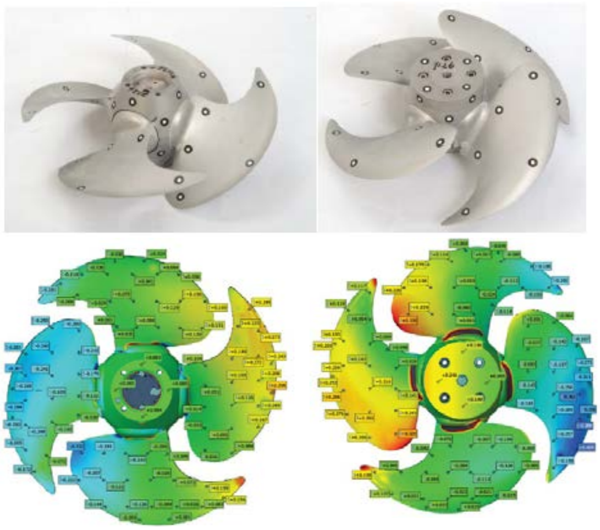


Figure 21: Optical scan on a CP propeller (deviation from its theoretical geometry, pressure side –suction side) (Dang *et al.*, 2012)

2.2.3 Non-stationary blade force measurements

Non-stationary blade forces measurement on propellers at model scale is still a challenging issue. Funeno *et al.* (2013) developed a blade spindle torque sensor built in the propeller hub (Figure 22) to measure blade torque of a controllable pitch propeller operating in off-design conditions and high propeller loading.

Just to mention that there is an increase interest (DNV rules, 2010, 27th ITTC Specialist Committee on Hydrodynamic Noise) for this topic that might have some impact on propulsor design, on the procedure to measure propulsor radiated noise, on the prediction of the cavitation inception point, because cavitation is largely increasing the radiated noise of a propeller (Briancon *et al.*, 2013; Bosshers *et al.*, 2013).

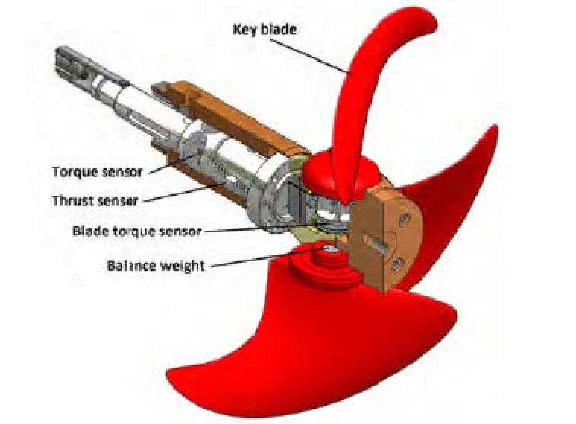


Figure 22: Propeller shaft thrust and torque sensor and the blade spindle sensor (Hagesteijn, 2012; Funeno, 2013)

2.3 New benchmark data

A sensitivity study of the testing parameters for Propeller alone and the Podded propulsor open water test have been carried out by CTO (Głodowski *et al.*, 2013). The paper is a summary of the large test campaign that was performed within the framework of the Hydro-Testing Alliance Network of Excellence, Joint Research Program 4 (JRP4), based on the so-call ABB case. Before that, a first benchmark had shown large discrepancies of about 5.9% even for the POD propeller open water test (Veikonheimo, 2006). Then a second benchmark testing program has been launched through the Hydro Testing Alliance (HTA) European Project to standardize the testing procedure in order to understand the causes the discrepancies found in the first test and to define recommendations for the testing procedure/setup of Podded propulsors. It was determined that using a same propeller model, a same POD housing, the same aft fairing for the propeller open water test, reduces the discrepancies of the results between the different facilities.

The final conclusions of this benchmarking test program were in line with the recommendations given in the 7.5-02-03-01.3 Propulsion, Performance Podded Propulsion Tests and Extrapolation. The authors recommend having a aft fairing cone to rotate with the propeller and having a separate pre-test with a dummy hub to correct with the propeller open water test results which is a first alternative recommended in the 7.5-02-03-01.3 Propulsion, Performance Podded Propulsion Tests and Extrapolation.

2.4 Application of computational methods

With respect to the propulsive performance, the major interest is still in developing and applying CFD (mainly RANS) models for self-propulsion simulation at model scale, including different approaches to extract the effective wake field. Meanwhile, such simulation at full scale began to appear, which provides a new perspective for studying the Reynolds scale effects. On the other hand, there is a pronounced increase in efforts devoted to the research of scale effects on energy saving devices, such as the pre-propeller stators/fins and ducts, the CLT propeller and the PBCF, and on multi-component propulsors, such as ducted, contra-rotating, and podded propellers.

2.4.1 Self-propulsion and effective wake field

In Castro, *et al*. (2011) the feasibility of self-propulsion simulation at full scale was demonstrated for the KCS, using a DES model and a dynamic overset approach. The propulsion factors were analyzed from simulated full scale resistance, open water, and self-propulsion performances, and compared with those obtained from model scale simulations and experiments. The SFC based on EFD and ITTC extrapolation procedure was larger than that based on model and full scale computation results. The computed full scale open water thrust was close to, while the torque was slightly lower than the model scale EFD data. The simulation results of full scale self-propulsion agreed well with EFD data, except for the torque coefficient (and hence the relative rotative efficiency). Through comparison of simulated stern flows, it was shown that the propeller working behind the hull experiences an inflow of higher axial velocity and uniformity due to the thinner boundary layer at full scale than at model scale, which results in favourable effects on propeller and propulsive efficiencies, axial loading fluctuations, as well as the level of bending moment around horizontal axis.

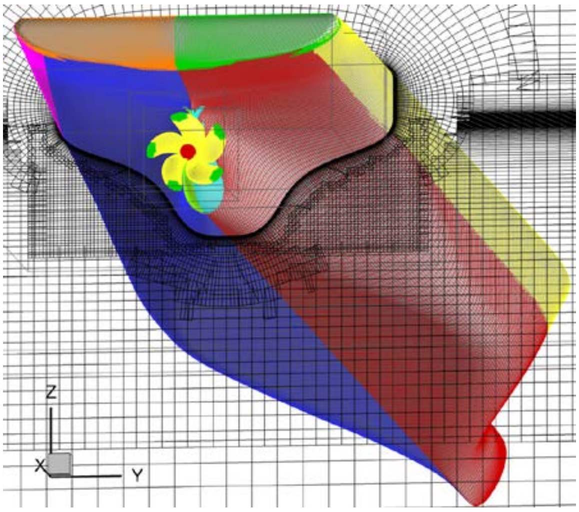


Figure 23: The overset grid system for KCS (Castro, *et al.*, 2011)

Villa, *et al*. (2012) presented a viscous/inviscid coupled approach for the simulation of self-propulsion based on a RANS and an unsteady BEM solver. Extraction of the effective wake field was made at a plane 0.2*D* upstream of propeller blades by using the time-averaged induced velocities computed by the BEM code. Simulations were conducted for the KCS propelled by KP505 at model scale, $F_n = 0.26$. It was shown that the iterative approach converged fast and the CFD and EFD results of total resistance, propeller rotation speed, and the velocities at a section in propeller slipstream correlated well. In addition the results from an actuator disk model (having axial force only) were also presented to compare with those from the more accurate BEM-based body force model.

Sakamoto, *et al*. (2013a) presented research on a RANS simulation of the resistance, open-water, and self-propulsion performances for a twin-skeg container ship at model scale, together with towing tank experiments. An in-house FVM solver and the Spalart-Allmaras one-equation turbulence model were used. The propeller was modelled by body force distributions computed by a simplified propeller theory. By using three sets of block-structured grids having a refinement ratio of $\sqrt{2}$, the uncertainty analysis for resistance and self-propulsion coefficients was conducted with the V&V method recommended by the 25th ITTC. It was concluded that the CFD solver was capable of predicting the resistance and self-propulsion performances for the low *L*/*B* twin-skeg ship, though it could be improved by implementing real-geometry propeller computations.

In Rijpkema, *et al.* (2013) different ways to extract the effective wake field and their influences on predicted propeller performance behind the hull were studied. Two in-house RANS solvers coupled with a BEM propeller code were used in a comparative investigation of the self-propulsion computations for KCS. The body force field was imposed at the blade positions (instead of the propeller disk position) by interpolation of BEM output. The effective wake field was obtained by subtracting the time-averaged propeller induced velocities computed by BEM from the RANS-computed velocity field of the hull-propeller system. The numerical results indicate that the effective inflow accelerates towards the propeller, see Figures 24 and 25, hence the axial location where the effective wake is defined has an influence upon the predicted propeller rotation rate.

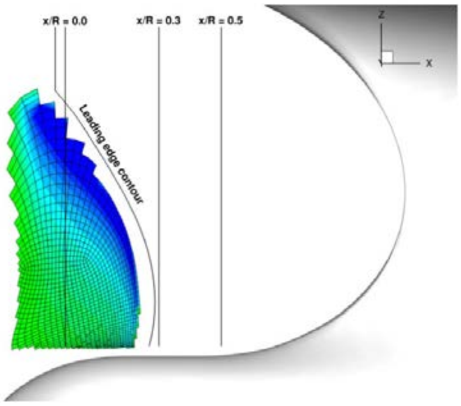


Figure 24: The axial locations used to compute the effective wake field. The contours represent the body force distribution in the RANS simulation. (Rijpkema, et al., 2013)

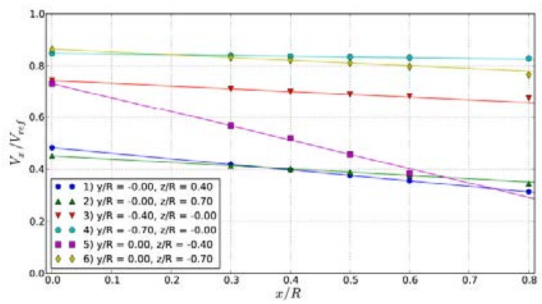


Figure 25: The RANS/BEM coupled results of effective wake velocities at different locations, where the acceleration of effective wake flow towards the propeller disk is shown. (Rijpkema *et al.*, 2013)

By using the effective wake field linearly extrapolated from upstream locations to the propeller reference plane, the accuracy of propeller performance prediction was improved. In addition the effective wake fields predicted by a RANS-BEM coupled method and by the more traditional force-field method based on the nominal wake and propeller thrust loading were shown to be quite different, being the former method more accurate with respect to the propeller performance when the extrapolated effective wake was used.

In the viscous/potential flow coupled approach by Sánchez-Caja, *et al.* (2014b) a lifting line model for the propeller in effective wake was used to compute the body forces which were circumferentially averaged and distributed on the propeller's reference plane, or the actuator disk. However, it was shown from open-water computations that the propeller-induced velocities by the lifting-line model were different (and not accurate due to the assumptions made in the model) from those predicted by RANS using the equivalent body force distributions, which would bring about errors in the effective wake so predicted.

For the three components of induced velocity vector, a procedure was proposed to quantify the correction factors for such errors due to the lifting line model through coupled computations for the open-water propeller. Numerical results indicated that the correction factors at a reference thrust loading condition could be applied, with just a little loss of accuracy, to another condition where the thrust loading was within ±50% of that at the reference condition. This feature might allow for savings in the computation of the correction factors. The procedure was applied to a hybrid CRP pod configuration, where it was shown that the errors in thrust and torque were about ±5% without corrections for the effective wake. As the interaction between forward and aft propellers was treated as part of the effect wake, the procedure would make it possible to use single propeller design methods for the CRP.

In the naval context, Liefvendahl, *et al.* (2012) presented near-wall modelled LES simulation results for the SUBOFF+E1619 configuration, using fine (16M) and coarse (8M) grids respectively. The authors found that the coarse grids resulted in a slightly higher level of unsteadiness in the wake flow, but a higher level of fluctuation in blade thrust, and concluded that much higher grid resolution would be needed for more accurate simulation of unsteady flow and propeller forces. Zhang *et al*. (2012) presented RANS simulation results for another submarine hull/propeller configuration to investigate the effects of free surface on resistance and self-propulsion. The CFD and EFD data correlated well, both indicating that the free surface effect on resistance was negligible below a certain centreline submergence ($h/L > 1/3$), while that on the self-propulsion factors was even smaller.

2.4.2 Energy-saving devices

The scale effects of the Wake Equalizing Duct (WED) and the Vortex Generator Fins (VGF) on propulsion and fluctuating pressure were numerically and experimentally investigated by Heinke *et al*. (2011). The RANS predictions of the nominal wake at model- and full-scale indicated that the propeller inflow would be largely altered by the scale effect, while the WED, or the VGF alike, could reduce the wake peak distinctly. The CFD results were utilized in designing the VGF. Concerning the scale effects on propulsion factors, discrepancies existed between the RANS and ITTC '78 predictions, especially for the WED, though the hull efficiencies happened to be close to each other.

Huang *et al*. (2012) presented an investigation of the WEDs for a bulk carrier based on RANS simulations with a body-force propeller model and experiments. Both work indicated that, for a fixed tilt angle, the asymmetric arrangement of port and starboard half-ducts was quite important for maximizing the energy-saving rate. The RANS-based energy-saving was slightly lower than the model test result.

The effects of symmetrical and asymmetrical WEDs for a VLCC were investigated by Yu *et al*. (2013) based on RANS simulations and a real-geometry propeller model. In this study, the extrapolated energy-saving rate from the RANS results was somewhat higher than that from the model tests.

In Guiard *et al*. (2013) the procedure for designing the Mewis Duct® was presented in brief. The fin setting designed on the basis of model experiments was subject to further adjustments to make full use of the full scale wake flow. In this final step of design, RANS simulation results at model and full scale provided the designer with a reference. Despite the lack of full scale wake data, it was assumed that existing procedure for scaling the effective wake fraction was applicable to scaling the nominal wake fraction, too. And it followed that the full scale nominal wake distribution would be deemed as a good prediction if its disk-average was close to the nominal wake fraction predicted by an accepted wake scaling procedure.

For a mid-size tanker, the influences of grid size, turbulence model, and surface roughness on predicted nominal wake flow and fraction were investigated. For a sufficiently fine grid set of high quality, it was shown that the nominal wake fractions were under-predicted with both SST k-ω and RST turbulence model if the hull was treated as a smooth surface. The surface roughness value was shown to have important impacts on the predicted wake distribution and fraction. In terms of the predicted wake fraction, the RST model performed the best for the typical roughness value of 0.188mm, while the SST k-ω model needed a much larger roughness of 0.5mm to yield similar result. Meanwhile, in the two cases the simulated wake flows were quite different (Figure 26).

A numerical investigation was made by Huang S.-Q, *et al.* (2012) for the effects of the Pre-Swirl Duct (PSD), a combination of a pre-positive duct and several pre-swirl stators. Four cases having different duct and/or stator section profiles were simulated. The device's effects of equalizing and pre-swirling the propeller inflow were confirmed by the RANS results. It was concluded that, among others, the stator pitch angle was a key parameter for the energy-saving predicted.

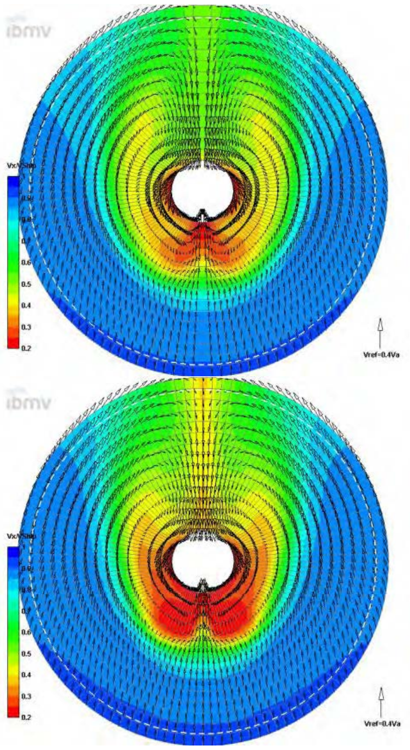


Figure 26: The full scale nominal wakes of a tanker predicted by RANS simulations. Upper: RST model, roughness 0.188mm; lower: SST k-ω model, roughness 0.5mm. The axial wake fractions were 0.313 and 0.318 respectively, against the value of 0.316 estimated by using the model wake fraction obtained from RANS and the RST model. (Guiard *et al*., 2013)

In Haimov *et al*. (2011) the scale effects on ducted and CLT propeller performance in open water were investigated. The RANS computations were conducted with a commercial solver, using unstructured tetrahedral grids. In respect to the CLT propeller, the comparison between model and full scale results indicated that the increase in efficiency at full scale was primarily due to the increase in thrust, in comparison to the decrease in torque. Meanwhile, the RANS-predicted scale effect was smaller when compared with a scaling procedure practically used for the CLT propeller. In addition, the RANS-predicted scale effects under lighter loading condition were more pronounced and much larger than for conventional propellers.

Sánchez-Caja, *et al*. (2014a) investigated the influences of endplate geometry on the efficiency of the CLT propeller based on full-scale simulations for two propellers in open water using an in-house RANS solver and Chien's low Reynolds number turbulence model. To reduce numerical uncertainty, a template-based procedure was devised for generating block-structured grids having the same topology and similar grid size distributions for different endplate geometries. The grid dependency was studied by using three successively coarsened grid sets. The largest difference was 1.3% in torque between the coarse and fine grids. Twelve cases were investigated for a 4-bladed propeller with varied endplate geometries. It was shown that the contraction of endplate affects both efficiency and thrust, and lighter loading on endplate improves the efficiency; the forward sweep improves the efficiency, too. From a theoretical viewpoint, the working mechanism of endplate was analyzed based on radial distributions of bound and free vortices obtained by integrating the flow velocities. The results indicated that for cases of higher efficiency the tip vortex was weaker, consequently the induced drag was smaller.

The RANS-based investigation was further conducted by Sánchez-Caja, *et al.* (2014c) of scale effects on the 4-bladed CLT propeller. The propeller efficiency at full scale was 10% higher than at model scale, where 2% was from the endplates due to the reduction in torque. The circulation distribution at full scale was higher in magnitude but lower in slope, confirming the results of higher thrust coefficient as well as efficiency than at model scale. The strong dependency of efficiency scaling on the type of flow regime at model scale was pointed out. In some cases, efficiency-based ranking for endplate designs at model scale were different from that at full scale.

2.4.3 Multi-component propulsors

Kinnas, *et al*. (2013) presented a method based on potential/viscous flow iteration by treating the duct-induced velocity field under propeller action as the effective wake for the propeller, for the purpose of computing propeller sheet cavitation under inclined inflow. The comparison of computed and measured open-water performances of a ducted propeller indicated that the iterative method was able to predict the hydrodynamic performance with good accuracy, especially for the thrust.

In Bulten, *et al*. (2011) a RANS approach was employed to predict and analyze the Reynolds scale effects on the open water performance of a ducted propeller and on the nominal wake of a ship hull. In regard to the ducted propeller, RANS results for the Kaplan 4-70 propeller in 19A nozzle indicated that the increase in open-water efficiency at full scale was mainly attributed to the decrease in propeller torque, while the duct and propeller thrust coefficients were quite close between model and full scales. An analysis was made to explain the results by employing the pump theory. It was argued that the reduced viscous loss at full scale had resulted in an increase in the dimensionless flow rate through the nozzle. Consequently the propeller loading was reduced at full scale, which resulted in decreases in both thrust and torque of the propeller, apart from the traditionally acknowledged scale effects on propeller thrust and torque.

The simulation approach was further extended for use with steerable thrusters in Bulten, *et al*. (2013). In this case the RANS-predicted model and full scale performances of the thruster in a straight course indicated that the efficiency increase at full scale was mainly due to the increase in unit thrust, in which the axial forces on the duct and propeller housing made more contributions. For the bollard pull performance, a generic prediction method was proposed by making use of the pump theory and RANS flow data, and the influence factors wherein were discussed. In respect to the transient thrust and torque, it was found from unsteady simulations that their fluctuation amplitudes were larger in free sailing condition than in bollard pull condition, and were asymmetrical about the steering angle. The reason for the asymmetry was further analyzed by comparing the contributions from the lateral force and the eccentricity of thrust.

To investigate the capability of the RANS simulation approach for ducted propeller under non-cavitating and cavitating conditions, CFD and EFD results were compared by Xia *et al*. (2012). The RNG k-e model with wall function, and Sauer and Schnerr's mass transfer model were employed for turbulence closure and cavitation, respectively. Block-structured and unstructured grids were used for fully wetted flow, where the predicted open water characteristics both agreed well with the measured one, except for lightly loaded conditions where the flow separation occurring near the duct trailing edge was not well simulated. Under developed cavitation conditions, although the numerical approach was able to simulate thrust breakdown, the thrust and torque were over-estimated in general and there were significant increases in predicted thrust and torque towards the starting point of thrust breakdown. It was concluded that the unstructured grids were more suitable for modelling the tip-clearance flow.

Sakamoto *et al*. (2013b) attempted model and full scale viscous CFD computation of a POD propulsor in open water configuration. The propeller was represented by a body-force model. The full scale propulsive efficiency estimated by the full scale CFD computation was higher than that of ITTC prediction method applied to the model scale CFD result (Figure 27). The discrepancy comes from differences in the resistance of the POD drive, caused by the changes of the flow field such as the position of the separation line.

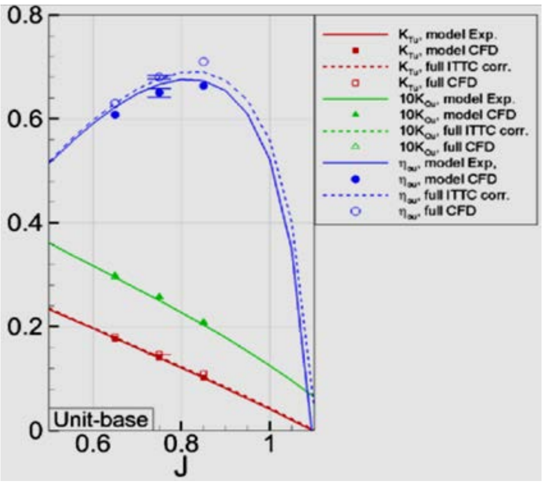


Figure 27 Comparison of propulsor open water characteristics: unit-based, Exp. vs CFD vs Exp. with ITTC correction, model and full scale. (Sakamoto *et al.*, 2013b)

Fujisawa (2013) discussed the scale effect on the POC of contra-rotating propeller by CFD. Fore and aft propellers of a CRP system have different trends in scale effect. It is supposed that the turbulence caused by the fore propeller hastens the flow transition of aft propeller (Figure 28).

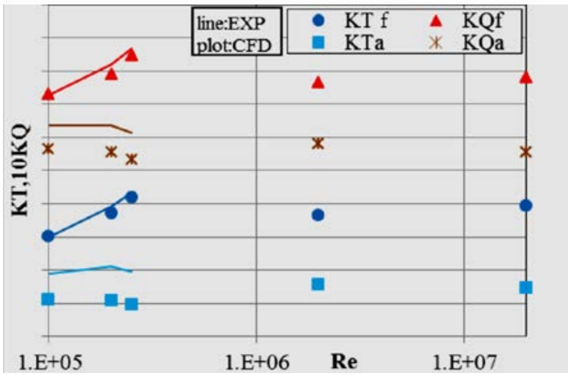


Figure 28: Scale effects on each of fore (subscript “f”) / aft (subscript “a”) propeller open performance (Fujisawa 2013)

In the ocean engineering context, thruster-hull interactions are important for DP system design. The numerical modelling of such interactions is challenging especially under the bollard condition. Maciel *et al*. (2013) presented a RANS-based approach to this problem and investigated its feasibility and accuracy in terms of predicted thrusts and wake flows for three typical cases, *i.e.*, a ducted thruster model working in open water, under a flat plate, and under a barge. The propeller was modelled by an actuator disk where the body force distributions were determined by fitting RANS results. By using a small current speed (J = 0.028) together with careful choice of numerical schemes and parameters, simulations under the bollard condition were realized with reasonably good agreement with the measured forces on the thruster and the plate/barge, and with the wake flows measured by PIV, see Figure 29 for example.

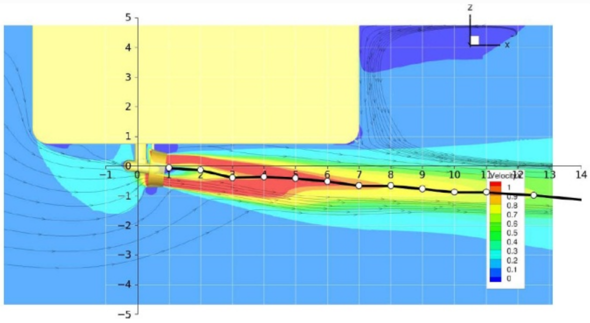


Figure 29: Comparison of RANS-simulated axial velocity contours with the wake trajectory measured by PIV for the tilted thruster working under the barge with bilge keel. (Maciel *et al*., 2013)

2.4.4 Design & optimization

Vesting *et al*. (2013) presented a multi-objective optimization study for a cruise ship propeller by considering the open-water efficiency as well as blade cavitation and induced pressures in a given wake field. A vortex lattice code was used to predict hydrodynamic forces and sheet cavitation, while a boundary element code to calculate the induced pressures. The codes were driven by a genetic algorithm. The blade geometry was parametrically represented in the traditional way, and morphed with B-spline curves in order to reduce the number of design variables. A sensitivity analysis (SA) was carried out for the geometric parameters with regard to their impacts on the performance aspects being considered. Starting from the baseline geometry which was already manually optimized and proven to be a good design by model experiments, further optimizations were conducted with all and a selected set (according to the results from SA) of the design variables, respectively. The latter case proved to converge faster than the former one. The results indicated that induced pressures could be further reduced. Satisfaction of the constant-K_T constraint, and the resolution for interaction effects among parameters remain as issues in the method.

2.4.5 Off-design operating conditions

Hur *et al*. (2011) measured propeller shaft torque and stress for an LPG carrier in a crash stop operation during sea trials. RANS simulations were carried out using the RPM and ship speed data from sea trial. Both steady and unsteady RANS results for torque were close to each other at full astern, and also to the sea trial data when the ship wake was ignored, suggesting that steady simulation could be used for blade strength analysis at initial design stage.

In Sileo, *et al*. (2011) RANS simulations were carried out for a self-propelled chemical tanker model under reversing condition, and the computed forces were compared well with measured data. Based on the numerical results of flow and hull pressure etc. the reasons for a reversing single-screw ship to deviate from straight course were analyzed.

The flow and forces under crash astern conditions were simulated by using LES for a single propeller, Jang, *et al*. (2013), and a ducted rotor with upstream stator rows, Jang, *et al*. (2012), and compared with model test data with good quantitative agreement. The typical flow feature was a highly unsteady vortex ring having its averaged location and strength changing with the advance ratio, Figure 30. Flow separation from the trailing edges resulted in high-amplitude, transient blade loads and the lateral force. For the ducted rotor-stator configuration, the same ring-vortex structure existed and numerical results indicated the lateral force came mostly from the pressures on duct inner surface due to tip-leakage flows. The LES model was further applied by Verma *et al*. (2012) to investigate the effect of hull on the propeller in crash astern operation. In addition to the ring vortex structure in the vicinity of blade tips, in the presence of the hull there existed a recirculation zone upstream of the propeller. The leading and/or trailing edge flow separation contributed to the transient lateral force.

In Amini *et al*. (2012) the feasibility of different numerical methods was investigated for predicting the six-component forces acting on the propeller blades of an azimuth pod thruster working in pulling and pushing modes within a ±30° range of the oblique inflow angle. The computational methods include a blade element momentum theory (BEMT), a boundary element method (BEM), and a RANS method. Being based on Glauert's momentum theory, the vortex cylinder model, and the blade element theory, the BEMT for oblique inflow was able to compute the blade forces and moments in a quasi-steady manner while the nominal wakes due to the hull and thruster housing were taking into account. Blade forces and moments obtained from the three methods were compared with those measured in a towing tank with a six-component force dynamometer embedded in the propeller hub. The MRF-based RANS model performed the best in both pulling and pushing modes, and the prediction accuracy was further improved when unsteady effects were accounted for by using sliding meshes. The two potential flow methods were able to predict the variations of blade forces and moments with oblique flow angle reasonably well, although under and over estimations were seen in the six components made either by the BEMT or by the BEM.

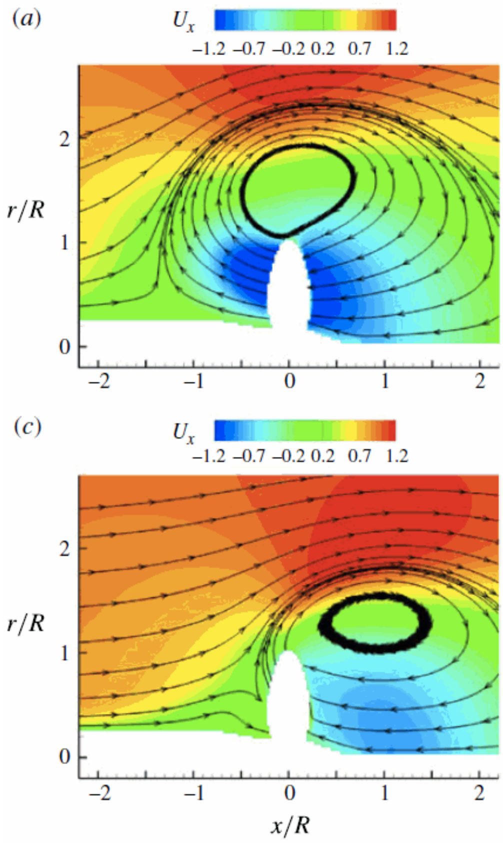


Figure 30: Circumferentially averaged flow around a propeller under crash astern operation. Upper: $J = -0.5$, lower: $J = -1.0$. (Jang, *et al*., 2013)

2.5 Experimental and CFD methods for the prediction of cavitation

Methods to predict cavitation on marine propeller blades has been classified by the 26th CFD Committee as interface tracking, discrete bubble dynamics and interface capturing methods. The interface tracking method is used to predict steady attached sheet cavitation in inviscid flow. In the discrete bubble dynamics method, cavitation is modelled as an interaction between bubble nuclei and pressure field variation. Bubble size governed by Rayleigh-Plesset Equation. This type of method is applied to predict inception, travelling bubble and nuclei effects. The interface capturing method assumes that the flow is a mixture of multi-phase flow and a flow solver and a cavitation model is used to determine the vapor volume of fraction.

Sipila and Siikonen (2012) have investigated the numerical simulation of cavitating model size PPTC propeller of SVA Potsdam in uniform inflow. The propeller wake field is calculated with Chien's k-ε turbulence model in the non-cavitating and cavitating conditions and with Menter's SST k-ω turbulence model in wetted conditions. The effect of the experimental coefficients in Merkle's mass transfer model on the cavitating tip vortex is studied systematically. The calculations are conducted with FINFLO, a general-purpose CFD code and numerical results are compared with model test results performed by SVA Potsdam. Particular attention was paid to the grid resolution. The numerical results and the experiments show a reasonable correlation with each other.

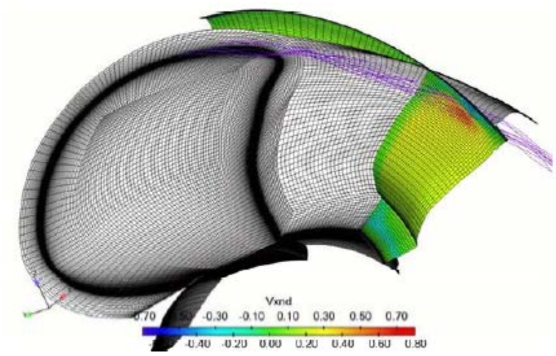
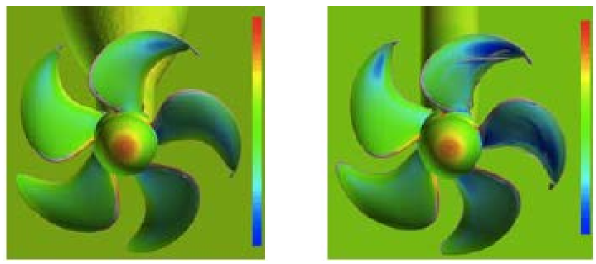


Figure 31 The surface grid on the blade and a slice of the grid in the slipstream that has been adjusted to follow the wake of the blade and the tip vortex at the finest grid level Sipila & Siikonen (2012).

Lu *et al.* (2012) performed numerical simulations of the cavitating flow around a typical yacht and Ro-Pax vessel propeller operating in open water but mounted on an inclined shaft. They used Large Eddy simulation (LES) and Unsteady Reynolds-Averaged Navier-Stokes (URANS) in combination with a Volume-of-Fluid implementation to capture the liquid-vapor interface and a transport equation-based method for the mass transfer between the phases. They compared the numerical results with the experiments. Their results indicate that a potential flow solver is not suitable for prediction complex sheet type, and root type of cavitation. RANS has partly captured the dynamic evolution of the sheet close to the tip region as well as the occurrence of the root cavitation. LES captured the correct location and dynamic behavior of the vortical structure (as was not the case for RANS) as shown in Figure 32. However the grid resolution is still an issue for the LES computation compared to those for the RANS.



(a) (b)

Figure 32: Blade pressure with iso-surface of the second invariant of the vorticity $\nabla\mathbf{v} - \nabla\times\mathbf{v}$, indicating vortical structures, as predicted by RANS (a) and LES (b)

Li *et al.* (2012) also made an attempt to predict numerically cavitating flows for the INSEAN propeller E779A operating in uniform and non-uniform wakes. A multiphase mixture flow RANS solver and Zwart's mass transfer model are used to predict the turbulent cavitating flow. Turbulence is modelled by a modified SST k-ω model. In the uniform wake, the predicted sheet cavities are stable and have similar patterns as observed in the experiment. They found that there are unresolved issues like the cavitation inception or disappearance leading edge cavity position, differences in the maximum cavity area and its location.

2.6 The need for R&D

There is still a need for continued R & D to aid in the improvement of model experiments, numerical modelling and full-scale measurement. Specific areas needing improvement are the following:

- model and full scale meaaurements of propulsors in off-design conditions
- full scale measurements of ship propulsive gain due to the use of Energy Savings Devices (ship configurations with and without)
- propulsive performances on composite propeller at full scale and model scale with possible measurement of blade deformation and torque
- full-scale measurements on Hybrid Contra-Rotating Shaft Pod propulsors
- EFD and CFD (e.g. RANS) simulation of the effect of varying Reynolds number on the performace of blade sections.
- full scale measurement of waterjet inlet flow velocity fields

2.7 High-speed marine vehicles

Performance prediction of high speed craft with a view to improve model/ship extrapolation techniques and additional investigations into scaling effects of waterjet and surface piercing propeller propulsion tests are focused on in this review.

2.7.1 Powering and performance prediction

Numerous studies are related to catamaran concept including well-known typical high-speed multi-hull model DELFT 372 catamaran for which new tests were also carried out and a large database is still in construction. Development of CFD tools for high-speed marine vehicles indicates its wide applicability in hull optimization processes and acceptable accuracy for power prediction.

Kandasamy *et al.* (2011) reported on hydrodynamic optimization of multihull ships. Simulation based design (SBD) was applied for the resistance optimization of two waterjet propelled high-speed ships, namely JHSS (Joint High Speed Ship) which is monohull and the DELFT catamaran. The adopted SBD explores the concept of variable physics approach for the Delft catamaran which shows strong waterjet hull interaction effects. The design optimization yielded geometries with significant resistance reduction for both JHSS and Delft catamaran. Tahara *et al.* (2011) also reported on the numerical optimization of the initial design of two waterjet propelled ships JHSS and Delft catamaran.

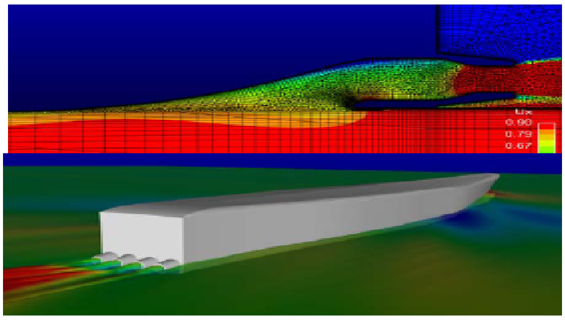


Figure 33: Axial velocity contours on a cross-cut inside the waterjet inlets (top figure) and aft view of the powered JHSS with water exiting the waterjet inlets and free surface colored by wave height (Delaney *et al.*, 2011).

Zaghi *et al.* (2011) reported on an experimental and numerical test campaign of fast catamarans being done at INSEAN facilities. The CFD models employed second order solver for the unsteady incompressible Navier Stokes equations. The effect of demi-hull separation by means of both experimental and CFD tools is reported. Delaney *et al.* (2011) performed RANS calculations on the JHSS that was equipped with four waterjets. Fig.33

indicates that the flow through the waterjet inlets is extremely complex including interaction with the free surface and these features captured correctly by CFD. Computed propulsion is within 6% of experimental measurements which is quite good considering slight differences between the computational model and experimental conditions.

Skejic *et al.* (2012) theoretically investigated the problem of effective power requirements in calm water for high-speed vessels (monohulls and catamarans) at preliminary design stage. The effective power requirements have been derived from a modified version of Doctor and Day (1997) method which predicts total vessel resistance in calm water, introduced modifications are mainly related to different methods of wave making resistance estimation for deep and shallow water. In particular for deep water the influence of the viscosity effects according to different wave theories is analyzed and demonstrated significant influence to effective power predictions. The wave making resistance and effective power are also analyzed in the finite water depth where they depend on the depth Froude number. The results are compared with available published results and show good agreement.

Eslamdoost *et al.* (2013) developed and validated the method, which is based on the potential flow assumption with non-linear free surface boundary conditions to model the waterjet-hull interaction. By means of this method, assuming that each of the investigated parameters independently influences the resistance change, the resistance increment of the hull is estimated through a linear expansion in a Taylor series, which is a function of the hull sinkage, trim and the flow rate through the waterjet unit. Knowing the magnitude of each single parameter separately helps to understand the physics behind the thrust deduction and may aid in the optimization of the hull/propulsor configuration. Also it sheds some light on the reason for the negative thrust deduction fractions sometimes found on waterjet driven hulls. Broglia *et al.* (2011) reported on calm water and seakeeping experimental investigation for a fast catamaran DELFT 372. The main issue of the paper is the interference effect between the hulls whilst former numerous studies published concerned with catamaran with the nominal separation. A monohull was tested as well. The large measurements collected provide a valuable database for CFD validation. Conclusions on the interference effects are made. The total coefficient curve shows the presence of two distinct humps one around Fr = 0.3 and one around Fr = 0.5. The peaks in the C_T are more accentuated for the catamaran than for monohull. Moreover it has seen that the second hump is strongly dependent on the separation length.

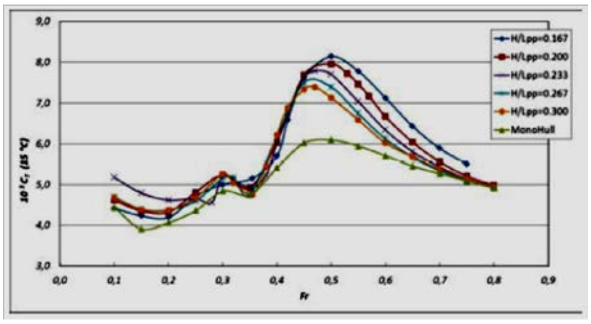


Figure 34: Calm water tests: C_T versus speed (Broglia, *et al.*, 2011)

Broglia, *et al.* (2012) provided the results of velocity field measurements around the Delft catamaran 372 model advancing in steady drift course. The purpose of the work is the characterization of the strong vortex structures generated along the keel of each demi-hull and to provide a valuable experimental data set for CFD benchmarking in severe off design conditions (such as during tight maneuver or when advancing at high drift angles). Zurcher, *et al.* (2013) discussed experimental set-up, model manufacturing and preparation for the model tests to be carried out using a load-varied self-propulsion testing technique and reported that a set of sea trials database is available to establish a methodology for waterjet self-propulsion testing based on the model tests.

2.7.2 High speed vehicle concepts

Besides widely used catamarans other concepts of high speed vehicles were also reported, including variations in their design by CFD and model experiment. Bono, *et al.* (2012) introduced the hybrid hull structure between a catamaran and a monohull (Y-shape) combining the positive characteristics of monohull (better maneuvering and less high frequency roll motions) with those of multihull (less resistance and good stability). Static and dynamic behaviors of the Y-hull model were studied in the towing tank experiment. Numerical simulations were carried out to optimize the hull shape. Brizzolara, *et al.* (2011) reported on hydrodynamic design of a family of hybrid SWATH unmanned surface vehicles. Modern CFD automatic parametric optimization has been developed as an instrument for design and the final design was validated by RANS approach. Eastgate, *et al.* (2011) discussed the design of a submersible aircraft concept along with the results of some tests.

McDonald, *et al.* (2011) reported on analyzing and comparing Tri-SWACH, monohull and trimaran concepts. Fu, *et al.* (2011) presented experimental and computational results for a Deep-V monohull planing hull. Planing craft model test program focused on collecting a wide range of types of measurements. Due to complexity of planning craft hydrodynamics model size was maximized (thus minimizing scaling errors) while still being able to obtain a wide Froude number range (0.31 to 2.5). Knight, *et al.* (2011) discussed the methodology of multi-objective particle swarm optimization of a planning craft with uncertainty. Zheng, *et al.* (2012) presented hull form design and performance evaluation of a Surface Planing Submersible Ship (SPS) which can sail in planning mode on the surface at high speeds and cruse underwater at low speeds. Finite volume based CFD method that took into account dynamic sinkage and trim were used for design of hull form. Preliminary model tests are also reported. Mosaad, *et al.* (2012) presented simple method to predict required power for WIG craft that can be used successfully in the preliminary design stage, using an iterative computer program. The output of this proposed method gives the logic and acceptable performance of the WIG craft compared to the related planning hull.

2.7.3 Propulsors

Hwang, *et al.* (2011) developed the design procedure for developing trans-velocity propellers (TVP). Trans-velocity (inflow-adopted) foil provided propellers design is that it jumps from non-cavitating condition to the super-cavitating at a very narrow speed range. Thus it operates like subcavitating propeller at low and intermediate speeds and transferred immediately to supercavitating mode at high speeds. Design procedure is effective but time consuming because of RANS. Boundary element method with cavitation model and viscous correction may be more practical for propeller geometry optimization with the RANS application at the last stage for final design. TVP is designed with efficiency 0.72 between 20 to 30 knots and 0.67 at 40 knots. Improvement of this TVP with relatively poor efficiency at high speed and large inclined shaft condition is demonstrated by Hsin, *et al.* (2013). The computational results from the RANS method are compared to the experimental data for both designs.

Propeller with hybrid sections (HB) *i.e.* different section geometries at different radii demonstrates better performance than TVP at all speeds and large shaft inclinations up to 10 degrees. A pre-swirl stator (PSS) is designed and as a result efficiency of HB improved by 1.38%. Figure 35 indicates final result of design optimization.

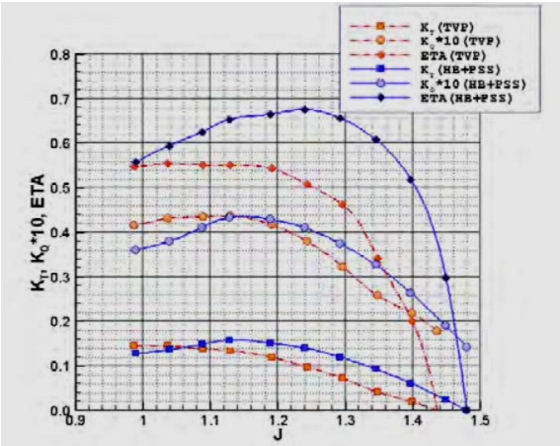


Figure 35: Comparison of performances of TVP and HB propeller with PSS at 36 knots and 10 degrees inclined shaft condition.

Epps, *et al.* (2011) represents the design method for high speed propeller blade shape optimization. During the optimization routine the design (ship endurance speed) load distribution is optimized, and the off-design (maximum speed) performance is determined, such that the chord length can be set to a minimum that still prevents cavitation at both conditions. Schulze (2011) and Weber (2011) discussed application of improved Z-drive with contra-rotating propellers for high-speed applications.

Dang, *et al.* (2013) reported on two new propeller series for Controllable Pitch Propellers (CPPs). Following the well knownWageningen *B*-series and *Ka*-series, the new *C*-series comprise open CPPs whereas the new *D*-series concern ducted CPP's. These series include 4-bladed CPP with large blade area and high pitch ratios for fast ferries. Systematic measurements of the propeller and duct thrusts, the torque and also the blade spindle torque have been carried out for the entire range of operational conditions and pitch-settings of each propeller.

2.7.4 Waterjets

Giles, *et al.* (2013) designed an advanced submerged type waterjet and reported on its hydrodynamic characteristics, including differences between powering performance predictions equipped with this propulsor estimated by the ITTC "momentum flux" method and by BMT's own method based on the proprietary software tool Ptool. Notable differences between the two methods were observed in the advanced waterjet in calculation of delivered power, with the empirical method giving a higher prediction then the momentum flux technique in the low to medium speed range due to higher estimated propulsive coefficients. The conventionally propelled hullform performance was derived from empirical estimates, with reasonably similar predictions throughout the speed range. The sensitivity study in the calculation of propulsive coefficients highlighted the need for further research to define a robust and mature calculation procedure for submerged waterjet technology.

Implementation of the ITTC recommended test procedures for waterjet systems has been discussed in Dang, *et al.* (2012) in detail by using a waterjet propelled 15 m Fast River Ferry as an example. Attention has been paid to scale effects of model testing and the method for Reynolds corrections. Self propulsion tests were conducted accordingly to ITTC procedure with the stock waterjet. Installed pump efficiency was found to be 4% lower than uniform free stream efficiency (due to flow distortion to the pump), although in most cases it is typically 2%. Determination of the uniform free stream efficiency may not be necessary if the pump efficiency in installed conditions can be measured correctly.

Significant attention has been paid to waterjet system efficiency tests. From the tests the pump head has been found to be less sensitive to Reynolds number if the flow at the pump inlet is fully turbulent, however impeller torque is highly Reynolds dependent. This means that a reliable waterjet system performance test should be carried out at high shaft rotational rates. Duct loss is strongly Reynolds dependent. Therefore, to get a good estimation of the duct loss, one must test at high duct Reynolds, a conclusion which is at odds with previous studies.

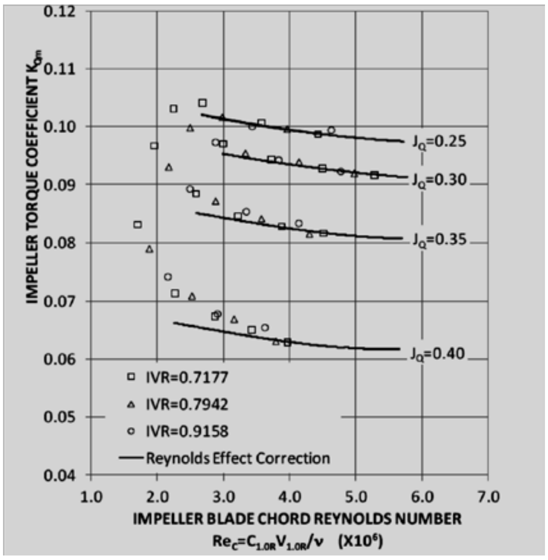


Figure 36 : Reynolds effect on impeller torque coefficients. (Dang *et al.*, 2012)

ITTC procedure required measurements to be conducted at more than one Reynolds number to get an appreciation of the Reynolds number dependency. Testing at blade tip chord Reynolds numbers as high as Rn = 5 x 10^6 is recommended by authors. Strong Reynolds dependency of the apparent intake losses have also been found which converges at a duct Rn = 10^7. A power loading coefficient is proposed for determination of the operating point when the pump characteristics are extrapolated to full scale while the existing ITTC procedure defined this point in terms of the towing force.

Chang *et al.* (2012) discussed a numerical panel method developed to predict the hydrodynamic performance of water-jets subject to a uniform flow. Steady potential flow inside the waterjet is calculated using a combined theoretical and numerical algorithm taking into account the interaction between the rotor and stator. The interaction between the rotor and the stator is evaluated using an iterative procedure that considers the effect of circumferentially averaged induced velocities from one rotor onto the other rotor. The pressures on the shroud surface inside the waterjet are evaluated by using hybrid scheme that couples the potential flow solver with RANS solver. Satisfactory correlations with the experimental data were observed. The predicted pressure head rise agreed well with experimental data and the maximum error is less than 2.5%. The predicted power coefficient is slightly lower (1% error) than those measured.

The predicted performance due to cavitation breakdown is well matched to the measurements. The current supercavitating model is to be improved and extended in order to analyze unsteady wetted and cavitating performance when the waterjet is subject to a non-uniform flow. The effect of air injection into a water jet is presented earlier by Tsai, *et al.* (2005) and then by Gany, *et al.* (2008), Gany (2011) was as much as 15 to30 % in terms of waterjet thrust. Gowing, *et al.* (2012) and Wu, *et al.* (2010) demonstrated details of test procedure development as well as optimization of the air injection waterjet. As a result Wu, *et al.* (2012) reported on net thrust augmentation as high as 70% (compare to 50% reported by Gany, *et al.* and 12% or 10% in Gowing and Tsai) for an exit void fraction of 50%. It is demonstrated that a well-designed nozzle with a proper air injection scheme can provide significant performance improvement with high void fraction air injection. The conclusions are based on numerical and experimental results

2.7.5 Surface Piercing Propellers

Surface Piercing Propellers (SPP) are often employed on high-speed vessels planning to reduce frictional resistance. Thus SPP's operate in fully or partly ventilated conditions, making SPP's difficult to design with high reliability. Himei, *et al.* (2013) discussed two theoretical methods for SPP analysis. One is modification of a program code for super-cavitating propellers using the vortex lattice method, and the other is RANS simulation applied the VOF method. The main conclusion was that analysis program for fully submerged super-cavitating propellers with correction of calculated Kt, Kq values by an immersion factor equal to the ratio of submerged propeller disc area to total propeller disc area could provide reasonable results in a short period of time. Also, RANS simulations showed good agreement with experiment, although the differences of both were larger at higher J (Figure 37). Scherer, *et al.* (2011) discussed theoretical and experimental results for surface piercing outboard and stern-drive propulsion systems.

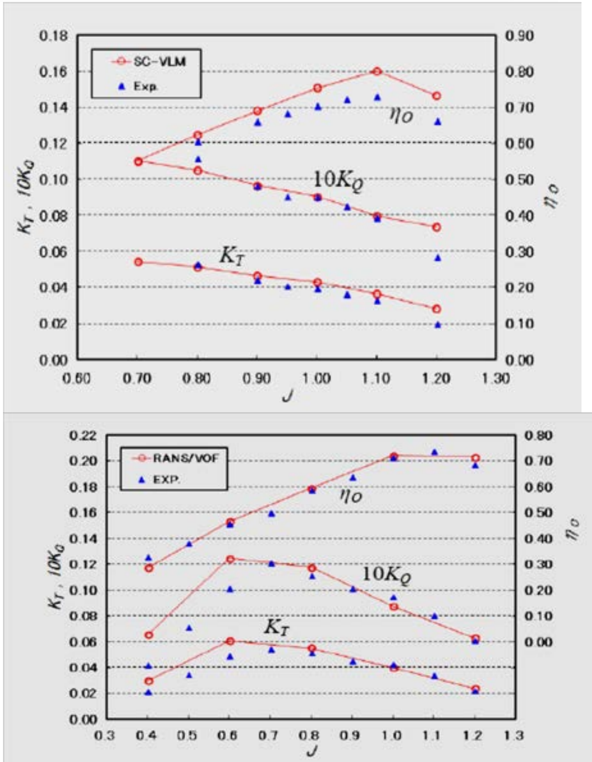


Figure 37: Comparison between calculated and measured K_T, K_Q and η_0. (Experiment: $F_{nD} = 6$, $\sigma = 2.3$) (Himei, *et al.*, 2013).

3 REVIEW ITTC RECOMMENDED PROCEDURES

3.1 Identify any required changes

The committee was given a task to review the procedures relevant to propulsion. In view of this the following procedures were subjected for reviewing:

- 7.5-02-03-01: Propulsion/Performance category including five sections.
- 7.5-02-03-02 Propulsion/Propulsor category including five sections.
- 7.5-02-03-03 Propulsion/Cavitation category including eight sections.
- 7.5-02-04 Ice Testing category including one section.
- 7.5-02-05 High Speed Marine Vehicle

category including one section.

- 7.5-02-07-02 Loads and Reesponses/Seakeeping category including one section.

After discussion with the Advisory Council the procedures for the Ice Testing, High Speed Marine Vehicles and Seakeeping categories were left due to a high number of procedures to be reviewed.

The following procedures were reviewed and updated by the 26th ITTC Propulsion Committee:

- ITTC Procedure 7.5-02 03-01.4 Performance, Propulsion 1978 ITTC Performance Prediction Method
- ITTC Procedure 7.5-02 03-02.3 Propulsor Nominal Wake Measurement by LDV Model Scale Experiments
- ITTC Procedure 7.5-02 03-03.2 Testing and extrapolation Methods Propulsion: Cavitation Description of Cavitation Appearances
- Update to ITTC Procedure 7.5-02 03-03.3 Cavitation Induced Pressure Fluctuations Model Scale experiments
- ITTC Procedure 7.5-02-03-03.4 Cavitation Induced Pressure Fluctuations: Numerical Prediction Methods
- ITTC Procedure 7.5-02-03-01.2 Propulsion, Performance Uncertainty Analysis, Example for Propulsion Test
- ITTC Procedure 7.5-02-03-02.1 Testing and Extrapolation Methods Propulsion, Propulsor Open Water Test.
- ITTC Procedure 7.5-02-03-02.2 Propulsion, Propulsor Uncertainty Analysis, Example for Open Water Test

Minor formatting corrections were made to Procedures 7.5-02-03-01.2 and 7.5-02-03-02.2 on uncertainty analysis as also made by the 26th ITTC Propulsion Committee. Procedure 7.5–02–03–01.4 on 1978 ITTC Performance Prediction Method was also modified for minor corrections and formatting. 7.5-02-03-01.5 Testing and Extrapolation Methods, Propulsion, Performance, Predicting Powering Margins has been effective since 25th ITTC, 2008 as a guideline. This procedure contains new terms, empirical formulae, etc. In Section 4.1.1 Calm Water Powering Margin, it is not clear how much power margin will be applied to the model tests results with either stock or design propellers. The committee thinks that the power margin requires validation and that a review of recommended power margins and power margin policy should be conducted, taking into account data for actual in-service performance in the typical service conditions encountered.

Thanks to a naval engineer, Mikael Huss who contacted the committee, the propulsive efficiency or quasi-propulsive coefficient, or total efficiency, η_D, equation was corrected in the 7.5-02-03-01.4 1978 ITTC Performance Prediction Method Procedure. In Section 2.4.3 the correct equation is $\eta_D = N_P \frac{P_E}{P_{DS}}$.

A small correction was made to the procedure 7.5-02-03-02.1, Open Water Test. In Section 3.1.1, Model, the procedure refers to propeller model accuracy as “The model propeller should be manufactured in accordance with Standard Procedure 7.5-01-02-02, Propeller Model Accuracy”. Actually the referred procedure is for cavitation tests not for propulsion and open water tests. The open water procedure should refer to 7.5-01-01-01, Ship Model procedure. Therefore this was corrected in the open water procedure. In addition 7.5-01-02-02, Model Manufacture, Propeller Model Accuracy is confusing for users. The committee recommends that this procedure should include all tolerances for model manufacture of propeller in two sections, including

Section 1 for propulsion and open water tests and Section 2 is for cavitation tests.

Some additional information on required calibration of LDV measurements has been included in the 7.5-02-03-02.3 Propulsor Nominal Wake Measurement by LDV Model Scale Experiments. As a matter of fact, to really follow a Quality System, the calibration of the calculation of the velocity $V=f_D \, . \, d_f$ should include both the calibration of the frequency processed by the Burst Signal Analyser and the calibration of the fringe spacing with the rotating disk. Then the velocity uncertainty can be determined using both the uncertainty on the Doppler frequency f_D and on the fringe spacing d_f :

$$\frac{\Delta V}{V} = \frac{\Delta f_D}{f_D} + \frac{\Delta d_f}{d_f}$$

Concerning the 7.5-02-03-03.3 Cavitation Induced Pressure Fluctuations Model Scale Experiments procedure, the committee recommends distinguishing two types of analysis: harmonic analysis *i.e.* blade angular position domain analysis [$p(\theta)=p(\omega.t)$], and time domain analysis [$p(t)$]. The first one is recommended to eliminate the potential fluctuation of the shaft revolution rate if the pressure is sampled using a multiple pulses shaft encoder. The second is preferred when examining broadband level of the pressure pulse signal.

Concerning the 7.5-02-03-03.4 Testing and Extrapolation Methods Propulsion; Cavitation Cavitation-Induced Pressure Fluctuations: Numerical Prediction Methods, the committee recommends to adopt the following revisions:

- Under Section 2.2.1, three references were added which reveal the most recent advances in effective wake calculation by the coupled RANS/potential-flow methods.
- Under Section 2.3.1, the status description for cavitation prediction was updated. The RANS and two-phase flow methods are now capable of predicting unsteady sheet cavitation though the accuracy needs to be improved, instead of being unable to address the problem.
- Under Section 2.4, a recent publication was added and commented to support the existing description of the more accurate but complicated method for hull pressure calculation.
- Under Section 3.1, “rake” was added as one of the propeller geometric parameters which are to be considered in pressure fluctuation computations.

Concerning the 7.5-02-03-03.2 Testing and Extrapolation Methods Propulsion; Cavitation Description of Cavitation Appearances procedure, the committee recommends providing sketches of propeller blade with the cavity extent on the suction side as a function of blade angular position (Figure 38).

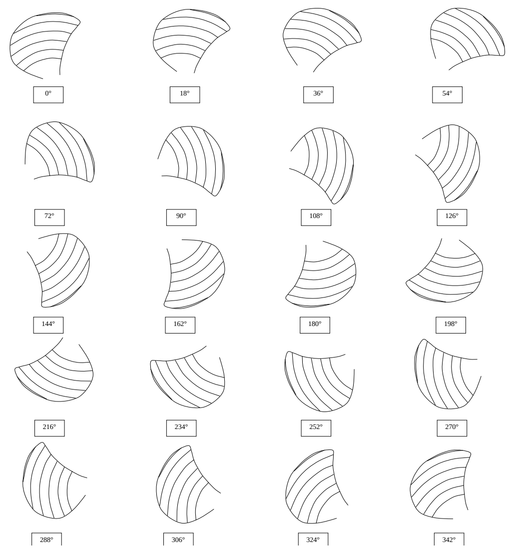
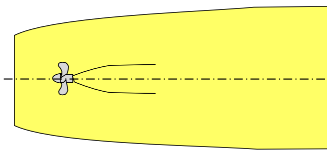
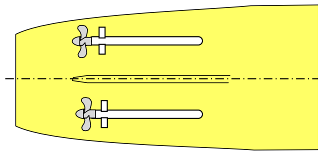
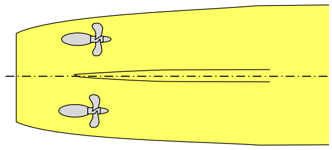
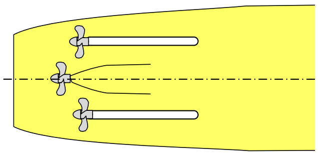
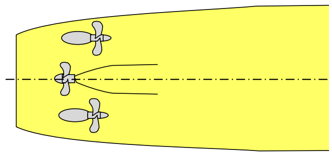
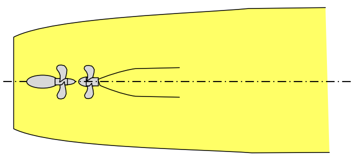
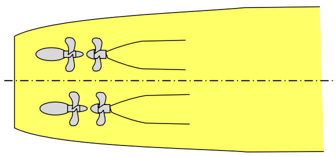
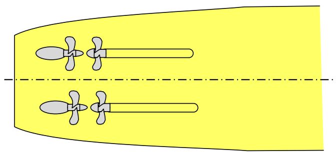
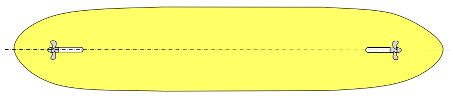
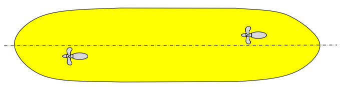
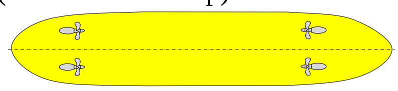
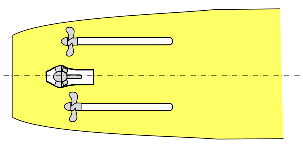


Figure 38: Suction side cavitation as a function of blade angular position

3.2 The need for new procedures

The committee discussed the need for new procedures and concluded that ships with multiple shafts and pre-swirl duct concepts with an integrated pre-swirl fin system are the main potential areas to be considered. Since the current procedures deal only with single and twin-screw propulsion, propulsion test with multiple shafts (mainly three) should be addressed and standardized for more accurate full-scale prediction. The committees recommend classification of existing propulsion systems as shown in the next table along with the existing or required self-propulsion procedure that should be applied for each class.

Table 1: Propulsion system classification

CASE I	Single shaft line Propeller 	Twin shaft lines Propellers or Pods  	Already existing ITTC self-propulsion procedures
CASE II	Center line Propeller + wing conventional shafl line propeller 	Center line Propeller + wing Pods / Thrusters / Z drives 	Need for self-propulsion procedure that should include differentiation of wake fraction and thrust deduction factor for wing and centre propellers and issues on power distribution. Possible extension of the existing procedure
CASE III	Single Shaft Line CRP Concept Conventional Propeller / Pod combination 	Twin shaft lines CRP Concept Conv. Propeller behind skeg / Pod Combination  Conv. Propeller open shaft / Pod Combination 	A new guideline is proposed by the present committee for Hybrid Contra-Rotating Shaft Pod Propulsors (HCRSP) Model Test.
CASE IV	Single Forward and aft propulsors (double ended ship)  	Twin Forward and aft propulsors (double ended ship)  Water jet(s) combined with conv. propeller / Pods 	Need for self-propulsion procedure that should include issues on power distribution optimization

Within this classification, CASE III, which is related to Hybrid Contra-Rotating Shaft Pod Propulsors, concerns propulsion systems where a high interaction between propulsors occurs. CASE II and CASE IV involve configurations where low interaction might exist between propulsors. Interaction between propulsors means that the loading of one propulsor is influenced by the loading of the other propulsors. CASE I is the only propulsion configuration where the interaction is assumed negligible.

The EEDI requirement of the IMO is forcing the ship industry to look for solutions, apart from the conventional ones, to satisfy the new requirements. Mewis ducts are one energy saving device which has been increasingly installed on ships. The application of the system requires on a ship-by-ship basis by the help of CFD techniques. Scaling of wake and the angles of the fin system are critical issues. Therefore a combination of CFD methods and experiments should be done in a coordinated way.

The committee highlights the following issues:

- Multiple shaft line (number of shaft >2) propulsors and the need to address the power distribution in the self-propulsion analysis
- Hybrid propulsion system procedures and guidelines
- Scaling issues on Energy Saving Devices

4 LIAISON WITH THE PERFORMANCE OF SHIPS IN SERVICE COMMITTEE

The IMO developed an Energy Efficiency Design Index (EEDI) which expressed the ratio of total CO2 emission from combustion of fuel, including propulsion and auxiliary engines and boilers, taking into account the carbon content of the fuels in question, with the transport work, calculated by multiplying the ship's capacity (dwt), as designed, with the ship's design speed measured at the maximum design load condition and at 75 per cent of the rated installed shaft power.

$$EEDI = \frac{CO_2\ \ emission}{transport\ \ work}$$

A simplified version of the EEDI formula is as follows:

$$\frac{\left(\prod_{j=1}^{n} f_j\right)\left(\sum_{i=1}^{nME} P_{MEi}.CF_{MEi}.SFC_{MEi}\right) + \left(P_{AE}.C_{FAE}.SFC_{AE}\right)}{f_i.f_c.Capacity.f_w.V_{ref}}$$

Where

f_j	: correction factor for ship specific design elements
f_i	: capacity facto
f_c	: cubic capacity factor
f_w	: weather factor
P_{MEi}	: Power of i^{th} Main Engine
CF_{MEi}	: Conversion factor from Power to CO_2 for fuel of i^{th} Main Engine
SCF_{MEi}	: Specific Fuel consumption for fuel of i^{th} Main Engine
P_{AE}	: Power of Auxiliary Engine
CF_{MEi}	: Conversion factor from Power to CO_2 for fuel of Auxiliary Engine
SCF_{MEi}	: Specific Fuel consumption for fuel of Auxiliary Engine
Capacity	: Measure of carrying power, eg deadweight for Tankers
V_{ref}	: Ship Design speed

In this formula, only the following items can be determined from model tests (or full size power trails): P_{MEi}, V_{ref} and f_w.

The determination of the speed power curve to determine P_{MEi}, V_{ref} and f_w are covered by the following procedures:

- Seakeeping Committee; 7.5-02-07-02.2 Testing and Extrapolation Methods Loads and Reponses, Seakeeping Prediction of Power Increase in Irregular Waves from Model Tests
- Specialist Committee on Performance of Ships in Service; 7.5-04-01-01.1, 7.5-04-01-01.2 Speed and Power Trials Parts 1 and 2.

Comments have been made and forwarded to the relevant committees, but too late for a response to be received within this session.

It is our understanding that the Specilaist Committee on Performance of Ships in Service has decided not to produce a procedure for the F_w component in the EEDI formulation.

This committee recommends that the issue of Power Margins for satisfactory performance and for safety should be reviewed jointly by the Propulsion and the Ships in Service Committee in the next session.

5 IDENTIFY WHERE CFD CAN SUPPORT EFD AND THE NEED FOR HYBRID CFD/EFD PROCEDURES

5.1 Status of relevant developments

Review of the papers published in major symposiums and journals during the period of the Committee indicates that there is a continuously growing interest in applying viscous CFD tools for predicting the hydrodynamic and cavitation performances of various marine propulsors and energy-saving devices, in particular the Reynolds scale effects. RANS methods are the most commonly used method; meanwhile, for enhanced resolution of the flow the DES and LES methods began to be applied to more complex configurations or more demanding operating conditions (Castro, *et al*., 2011; Jang, *et al*. 2012 & 2013). On some topics, such as the ship wake, ESDs, and propeller at crash astern, fully RANS or combined viscous/inviscid tools are being used as complements to model experiments by providing data that are difficult or impossible to measure. Concerning powering performance prediction, Verhulst (2012) expected that hybrid procedures would emerge based on suggestions that CFD could be a better tool than model experiments for predicting resistance scale effect when flow separation is severe at model scale, and for evaluating scale effect on wave-making resistance.

For designing a wake-adapted propeller or predicting its cavitation behaviour, the effective wake field is needed, which can be predicted from the model-scale nominal wake field by a scaling method, such as the RANS or the Sasajima method as recommended by the Specialist Committee on Wake Scaling of the 26th ITTC.

The effective wake field is not generally directly measured by model experiments (using e.g. LDV techniques). As already mentioned in Chapter 2.4.1 , there are a number of researches dedicated to predicting the effective wake field based on coupled viscous/potential-flow CFD methods. The hull flow with propeller in action is simulated by the RANS method, while the propeller is replaced by a body force distribution. The potential flow methods are employed to compute the propeller working in iteratively determined effective inflow, such as the boundary element methods (Rijpkema, *et al*., 2013; Krasilnikov, 2013) or the lifting line method

(Sánchez-Caja, *et al*., 2014b). The computed propeller loads are converted to a body force distribution that are distributed on the propeller disk (Krasilnikov, 2013; Sánchez-Caja, *et al*., 2014b) or in the actual fluid volume otherwise swept by the rotating blades (Rijpkema, *et al*., 2013). The effective wake distribution defined at the propeller disk is determined by extrapolation from the velocities at planes upstream of the propeller blades (Rijpkema, *et al*., 2013), or by deducting from the total wake velocities the RANS-computed induced velocities for the actuator disk (without hull) in open water (Krasilnikov, 2013; Sánchez-Caja, *et al*., 2014b).

Potentially these methods are also applicable to full-scale. However, it is not yet clear how the different potential-flow and body-force models for propeller influence the predicted effective wake distribution. A comparative/benchmark study might be necessary to further assess the methods.

In ITTC recommended procedure 7.5-02-03-02.5: “Experimental Wake Scaling”, existing approaches have been listed for simulating the full scale wake. RANS simulations are recommended to help with the experimental simulation work, specifically in the scaling approaches using flow liners, water speed adjustment, and smart dummy models. In fact, the geometric particulars and even shape of the flow liner or smart dummy model are so designed that their nominal wake distributions according to RANS simulation best approximate the target ones which again are sometimes predicted by RANS. Hence the procedure is already a partially hybrid one. Further work based on viscous CFD seems necessary to investigate how the propulsor interacts with the wake simulating devices.

One category of energy-saving devices consist of pre-propeller fins, ducts, and the combination of them. They are designed to produce pre-swirl inflow to the propeller by making use of the swirling flow due to the bilge vortices, and/or to accelerate the high axial wake region. As the direction and speed of stern flow are strongly influenced by the viscous flow around the hull, scale effects are important for the design and performance extrapolation of such devices. Being a typical example, Guiard, *et al*. (2013) presented a design procedure for the Mewis Duct® where the fin setting designed according to model experiments might be subject to final adjustments based on full-scale RANS simulation results. Due to the lack of validation data, the full-scale simulation model was fine tuned, interpretation of the results and the final adjustments were made with care.

It has been known that the CLT and Kappel type propellers, as well as the propeller boss cap fins, are subject to more severe scale effect, and the ITTC'78 procedure originally designed for conventional propellers might be no longer applicable to them. RANS simulations seem to become a routine for the assessment of their scale effects. Besides, RANS tools are widely used in the design process for ESDs, see Section 2.4.2 and Section 6 for a comprehensive review, since it would be difficult and cost inefficient to improve the design by measuring the forces that are usually quite small, or by visualizing the flow experimentally.

For propulsors involving stationary parts, such as the duct and pod housing, scaling is an issue as the blades and stationary parts experience different flow regimes, which is further complicated by the change in load shares among the parts at full scale.

For ducted propellers, based on a RANS simulation and analysis, it was shown that the reduced viscous loss at full scale had resulted in an increase in the dimensionless flow rate through the duct. Consequently the propeller loading was reduced at full scale, resulting in decreases in both thrust and torque of the propeller, apart from the traditionally acknowledged scale effects on propeller thrust and torque (Bulten, *et al*., 2011).

The existing ITTC procedure for podded propulsor extrapolation still calls for validation by full-scale data. In practice viscous CFD tools are mainly employed to assess the scale effects. In a recent work a scaling procedure for puller pods was proposed, featuring a correlation coefficient, β, which is dependent on the Reynolds number as well as the thrust-loading coefficient. The correlation coefficient was evaluated with resort to full-scale RANS simulations (Park *et al*., 2013).

For flexible propellers, the practical difficulties were pointed out in Section 2.1.3 to satisfy both hydro and structural dynamic similarity laws. Instead, numerical simulation involving coupled fluid and structure analysis was recommended.

5.2 Needs for hybrid procedures

Based on the brief review in the preceding section, the Committee finds that, although CFD is being increasingly used to various aspects of ship propulsion, especially concerning the scale effects and helping with design by providing complementary data to experiments, it is still too early to recommend a new hybrid procedure mainly for two reasons.

First, the case studies available in the open domain are based on different modelling approaches which brings about many options and makes it difficult to judge their applicable extent. In this sense benchmark studies are necessary. Second, before a numerical approach can be incorporated into an existing procedure, it needs validation by full-scale data.

However, the Committee recommends that CFD should be gradually integrated into the overall tool set for making predictions in the same way as any other experience or theory based method is at present. Potential combinations of CFD and EFD are listed below,

- CFD-aided scaling of resistance and powering
- CFD-aided simulation of full scale and effective wake field
- CFD-aided performance scaling for ducted propellers, podded propulsors, and energy-saving devices
- Numerical scaling for flexible propellers

Another potential use of CFD is that calibrated CFD can be used to extend EFD results for items not measured, such as stern flow direction and the side force on propellers, and to give guidance in design modification.

6 MODELLING AND SCALING OF UNCONVENTIONAL PROPULSION AND WAKE IMPROVING DEVICES

Energy saving devices have become an important issue in recent times due to the increased price of oil and EEDI regulations. Many energy saving devices have appeared, however only some of these devices remain after validation of the effectiveness of their performance. In this study, these unconventional devices are classified and assessed in terms of their energy saving potential. The unconventional devices are classified into four categories, mostly based upon Carlton's (2012) criteria, as shown in Table 2. The maximum possible gains in the propulsive efficiency in model tests were recently shown by the HSVA group and are shown in Table 3. The most difficult thing about the comparison of the efficiency gains is choosing the basis case against which they are assessed, which could vary by up to 3% or more.

PRI give a pre-rotation to the propeller inflow
IPI improve propeller inflow
AFS alleviate flow separation
RRE recover rotational energy from downstream
DVP decrease viscous loss after propeller cap
DEP decrease eddy after propeller cap
DTV decrease tip vortex loss
PAT produce additional thrust

Table 2 Classification of unconventional propulsors based on Carlton (2012)

Devices before the propeller	Ducted propeller	Mitsui integrated duct: PAT, IPI
		Hitachi zosen nozzle: PAT, IPI
	Wake Equalizing Duct (partial duct)	Schneekluth duct: IPI, PAT
		Sumitomo duct:PAT, IPI
	Pre-swirl stator	Reaction fin: PRI
		Asymmetric stator (DSME): PRI
	Pre-swirl duct (Mewis duct): PRI, PAT, IPI	
	Flow regulating front fin	Grothues spoilers: IPI, PAT
		Saver fin (SHI): IPI, AFS
Devices at the propeller	Unconventional tip shape propeller	Propellers with End-Plates (CLT): DTV
		Backward rake tip propeller: DTV
		Forward rake tip propeller(KAPPEL): DTV
	Propeller Cone Fins (PBCF): DEP	
	Contra-Rotating Propeller: RRE	
	Rim driven (Hubless propeller): DTV	
Devices behind the propeller	Grim Vane Wheel: RRE	
	Rudder-Bulb Fins system: DVP	
	Additional thrustor fins: RRE	
	Post-swirl stator: RRE	
Renewable energy propulsion	Sail	
	Kite	
	Magnus effect	
Others	Oscillating propulsor	
	Surface piercing propeller	

Table 3 Maximum possible gains from measures aimed at increasing the propulsive efficiency (by Hollenbach, and Friesch, in HSVA)

		Possible Gain	**Model Tests required?**
Reducing Separations,/ Improving the Quality of the Wake Field			
Grothues wake equaling spoiler		3%	Yes
Schneekluth wake equaling duct		4%	Yes
Sumitomo integrated Lammeren duct	(SILD)	6%	Yes
Recovering Rotation Losses			
Twist rudder without rudder bulb	(BMS / HSVA)	2%	Yes
Single pre-swirl fin	(Peters / Mewis)	3%	Yes
Pre-swirl fin system	(DSME, Korea)	4%	Yes
Rudder thrust fins	(HHI, Korea)	4%	Yes
Reducing Hub Vortex Losses			
Divergent propeller boss cap		2%	Yes
Rudder with rudder bulb		2%	Yes
Propeller boss cap fins	(PBCF)	3%	Proposed
Reducing Rotational and Hub Vortex Losses			
Twist rudder with rudder bulb	(BMS / HSVA)	4%	Yes
High Efficiency Rudders	(Wartsila, Rolls Royce)	6%	Yes
Note: Possible gains are not fully cumulative			

6.1 Devices before the propeller

6.1.1 Ducted propeller

Recently, the conventional duct has been modified into several configurations. A partial duct is more popularly used than a conventional duct for equalizing the oncoming flow into the propeller as well as for saving energy. The Mitsui integrated duct and the Hitachi Zosen duct might not be classified as partial ducts but rather as a classical ducted propeller, because the size is almost the same as for a conventional duct, although the positioning and shape have been changed slightly (see Figures 39 and 40).

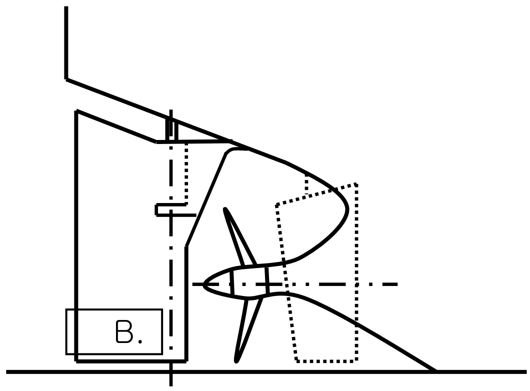


Figure 39: Mitsui integrated ducted propeller (Carlton, *et al.*, 2010)

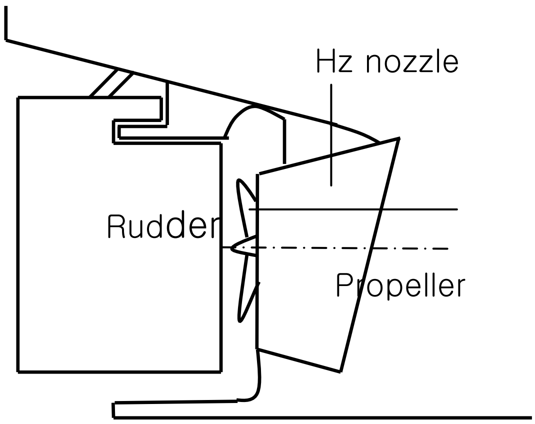


Figure 40 : Hitachi Zosen nozzle propeller (Carlton *et al.*, 2010)

Sumitomo's SILD has been successfully applied to a tanker, where the efficiency gain was more than 6% as shown in Table 4. There may be further room to improve the efficiency at slow speed ship with an optimized duct. The scaling problem for these ducts may differ somewhat from a conventional duct. Further work on this scaling problem, including pre-swirl in the duct is expected in the near future.

A EFD case study for the effect of a partial wake equalizing duct was carried out for a river-going general cargo ship by Korkut (2006). Analysis of the results indicates that the partial wake equalizing duct concept with an appropriate stern design affects not only the flow characteristics at the aft-end, but also the propulsion characteristics.

Bulten (2011) proposed a full scale numerical towing tank and wake field for analyses. The duct was analysed as an axial pump instead of using conventional propulsive coefficients within CFD. The well-known 19A nozzle and Kaplan type propeller were used to investigate scale effects as shown in Table 5.

Table 4 Typical efficiency gains from PIDs (Propulsion Improving Devices) from HSVA (Hollenbach & Reinholz, 2011))

Year	**Ship Type**	**Device**	**Gain in Power**	
			Design Draught	**Ballast Draught**
2010	ConRo Vessel	DSME Pre-Swirl Stator	3.7%	Not investigated
2009	Kamsarmax Bulk Carrier	DSME Pre-Swirl Stator	6.3%	1.4%
2009	7,450 TEU	DSME Pre-Swirl Stator	3.6%	Not investigated
2008	16,000 TEU	DSME Pre-Swirl Stator	3.8%	Not investigated
2008	13,050 TEU	DSME Pre-Swirl Stator	4.5%	3.2%
2008	14,000 TEU	DSME Pre-Swirl Stator	4.5%	4.7%
2008	4,400 TEU	DSME Pre-Swirl Stator	1.0%	Not investigated
2008	7,090 TEU	DSME Pre-Swirl Stator	3.3%	0.4%
2007	VLCC	DSME Pre-Swirl Stator	5.6%	5.5%
2007	6,300 TEU	DSME Pre-Swirl Stator	3.3%	Not investigated
2007	8,400 TEU	DSME Pre-Swirl Stator	3.5%	1.1%
2005	VLCC	DSME Pre-Swirl Stator	4.8%	Not investigated
2011	158k DWT Tanker	SHI Saver Fins	3.2%	Not investigated
2007	8,000 TEU	SHI Post Stator	3.9%*	Not investigated
2005	8,000 TEU	HHI Thrust Fin	4.9%	Not investigated
2007	Aframax Tanker	Sumitomo SILD	8.7%	Not investigated
2003	Aframax Tanker	Sumitomo SILD	6.0%	Not investigated

*measured in HSVA's large cavitation tunnel HYKAT at higher Reynolds Numbers

The research shows that the difference between the model and full-scale torque is larger for the ducted propeller case than for a conventional propeller for the same thrust. This paper also shows the possibility of full-scale wake predictions using CFD methods for the ducted propeller.

Table 5 Wake fraction comparison for the ducted propeller

	Measured	CFD (model scale)	CFD(full scale)
Wake fraction	0.5132	0.5041	0.3477

Heinke *et al.* (2011) investigated scale effects for ships with a wake equalizing duct or with vortex generating fins. The CFD calculations at the model and full scale show that the change in the propulsion coefficients, such as the thrust deduction fraction, wake fraction and hull efficiency of ships with a WED or VGF can be predicted with good accuracy using the ITTC 1978 propulsion method. The CFD calculations show a larger scale effect on the effective wake fraction than the prediction using the ITTC 1978 method for the WED.

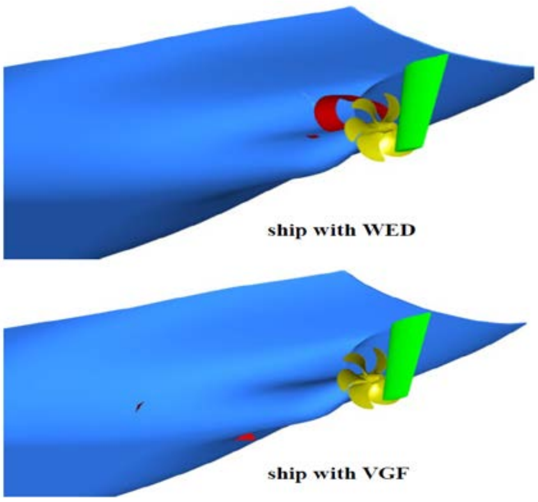


Figure 41 : Appendage profiles (Heinke *et al.*, 2011)

Analysis of the cavitation observations showed that tip vortex cavitation is only weakly developed if the propeller works in the full-scale wake field (DM69S). This effect could be an indication of the investigations which are necessary to in order to understand the impact of scale effects on the wake field and cavitation, in particular the tip vortex cavitation (Figure 42).

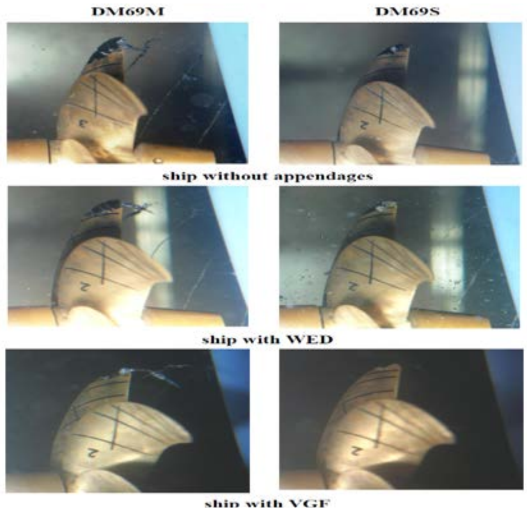


Figure 42: Comparison of Cavity extent (Heinke *et al.*, 2011)

Yasuhiko *et al.* (2011) investigated full-scale design of a semi-circular duct using CFD. The flow field at the front of the duct was analysed at both model and full-scale. From the change in the orientation of the vortices, the angle and diameter of the full-scale duct were changed.

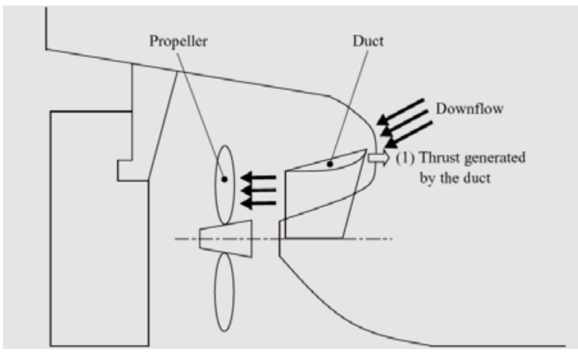


Figure 43: Basic energy-saving principles of the semi-circular duct (Yasuhiko *et al.* (2011))

6.1.2 Pre-swirl stator

A reaction fin has been successfully applied to high block coefficient ships by Takekuma *et al.* (1981) of Mitsubishi heavy industry. Research into the pre-swirl stator (see Figure 44) was extended to the development of an asymmetric stator by DSME (Daewoo Ship and Marine Engineering) and also the combination of a stator and wake equalizing duct.

Figure 44: DSME asymmetric pre-swirl stator (Source unknown)

The model test results are normally scaled using ITTC1999 method (Van *et al.* 1999) that differs from the ITTC1978 method in the prediction of full scale wake as shown in the equation below.

$$W_S = (t_{MO} + 0.04) + (W_{MO} - t_{MO} - 0.04)\frac{C_{FS} + C_A}{C_{FM}} + (W_{MS} - W_{MO})$$

where :

W_S = *ship wake*

W_{MO} = model *wake w/ostator*

W_{MS} = model *wake w/ stator*

t_{MO} = model *thrust deduction w/ostator*

$$W_S = (t + 0.04) + \frac{(W_M - t - 0.04)((1+k)C_{FS} + \Delta C_F}{(1+k)C_{FM}}$$

Although the wake scaling in the 1999 method may be somewhat exaggerated as a result of the axial velocity retardation due to the existence of the stator, there is a good correlation between the analysis result and sea trial. This may act as some compensation for having no scaling of the stator drag at full scale.

The new type of stator, the so-called L-J duct and the pre-swirl stator were recently introduced by Zondervan *et al.* (2011) as shown in Figure 45.

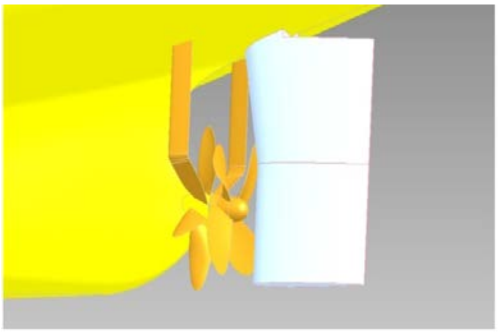


Figure 45: Illustration of a Bulk carrier fitted with an L-J duct and pre-swirl stator (Zondervan *et al.*, 2011)

A similar concept was applied to a twin shaft vessel to increase the efficiency through the use of struts. There have been many attempts at developing different configurations of pre-swirl stator.

6.1.3 Pre-swirl duct

Mewis developed a combination of a partial wake equalizing duct and an asymmetric stator that has a very compact configuration from a structural point of view. The model test results were published by HSVA model basin as shown in Table 6 where the efficiency gain was about from 2 to 7 percent compared to the conventional propeller. Manoeuvrability and cavitation tests were also conducted to compare the performance both with and without the Mewis duct. The Mewis duct has mostly been applied to high block coefficient ships such as bulk carriers and tankers as the duct is more effective.

The pre-swirl duct was analysed with a varying stator angle and angle of attack of the duct by Huang *et al.* (2012). The changes in the wake field were with the variation of parameters in a pre-swirl duct. An efficiency gain of between 2.9% and 4.3% was calculated using CFD computations of these variations.

Guiard *et al.* (2013) proposed a full-scale design for a pre-swirl duct through the use of a CFD code. It was stated that a complicated turbulence model is difficult to apply to a compound propulsor such as a Mewis duct. The computed results are expected to be validated by full-scale test data.

Table 6: Model test results with a BMS Mewis Duct from HSVA (Hollenbach and Reinholz, 2011)

Year	Ship Type	Gain in Power	
		Design Draught	Ballast Draught
2011	151k DWT Tanker	4.7%	Not investigated
2010	75k DWT Tanker	3.9%	7.2%
2010	163k DWT Tanker	4.7%	7.1%
2010	158k DWT Tanker	3.8%	Not investigated
2010	57k DWT Tanker	5.4%	7.8%
2010	20,000 DWT MPC	1.5%	Not investigated
2009	45k DWT Bulker	6.0%	5.4% *
2008	12,000 DWT MPC	7.7%	7.4%
2008	Aframax Bulk Carrier	6.9%	Not investigated

* light loaded draught condition

6.2 Devices at the propeller

6.2.1 Unconventional tip shape propellers

Sistemar Company has proposed the Tip Vortex Free propeller (TVF), though the name has subsequently been changed to the Contracted Loaded Tip propeller (CLT). The concept behind the design is the same as for a winglet on an airplane. This idea has been extended to both forward (so called KAPPEL) and backward smoothly curved tip rake propellers to mitigate cavitation risks at the propeller tip. The reduced strength of the tip vortex reduces the pressure fluctuations on the hull surface rather than improving the efficiency. The backward tip rake propeller has been applied to most propellers in Korean ship-yards recently.

Anderson (1997) reported three kinds of extrapolation for tip fins, using a method based on the ITTC78 method. To secure a fair comparison, this procedure was applied to both the tip fin and conventional propellers. The corrections turned out to be bigger for the tip fin propeller, meaning that it is more sensitive to scale effects. Unfortunately, no full-scale tests have been undertaken and so no confirmation of this scaling procedure can be made.

Inuakai (2013) conducted a comparative study on the performance of backward and forward (KAPPEL type) tip rake propellers. It was found that the negative pressure area on the blade can be significantly reduced with backward tip rake propellers. This means that the blade area can be reduced without sacrificing cavitation performance, which consequently leads to a 2.6% higher efficiency when compared with a conventional propeller.

Figure 46: Forward tip rake propeller (Source unknown)

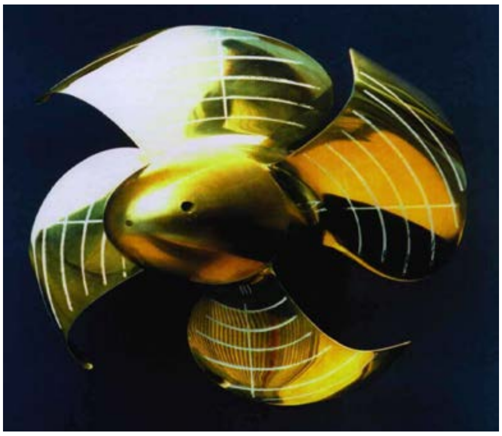


Figure 47: Backward tip rake propeller (Source unknown)

Figure 48: Contracted loaded tip propeller (Source unknown)

Bertetta (2012) carried out EFD and CFD work to analyse the CLT propeller. A panel method and RANS code were used in the computational analysis of the POW performance and cavitation. There was good agreement between the experiments and computations, with an overall error of less than 4%.

Inukai (2011) proposed the concept of a CRP with a tip rake propeller to improve the propulsive efficiency. The efficiency of the backward tip rake type is slightly better than that of the forward type but there was no gain found in the experiment. Ultimately, a 1.5% efficiency gain was found compared to a conventional CRP.

Sanchez-Caja et al. (2012) analysed the scale effects of the CLT propeller using CFD (RANS Solver). According to their computational results, the difference between the full and model scale is larger than for a conventional propeller because the flow separation area is somewhat larger than that of a conventional propeller. This means that the standard extrapolation method for a conventional propeller may not be applicable to a CLT type propeller.

Cheng *et al.* (2010) reported on the scale effects for an end plate effect propeller (KAPPEL) using both numerical computations and experiments. The CFD showed larger scale effect for both thrust and torque as compared with EFD, which only showed scale effect on torque.

Nielson *et al.* (2012) proposed a combined system of a KAPPEL propeller and rudder bulb whose efficiency was up to 9% higher than a conventional propeller system. The proportion of this gain from the KAPPEL propeller and rudder bulb were almost even.

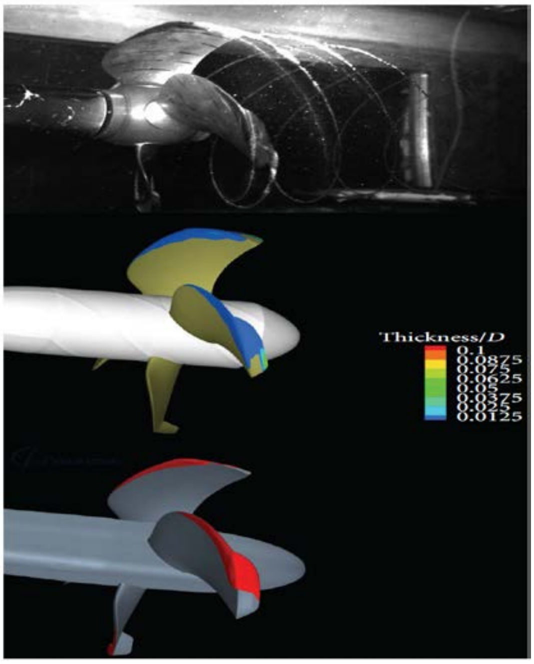


Figure 49: Comparison of cavity extent using panel and RANSE numerical computations (Cheng *et al.*, 2010)

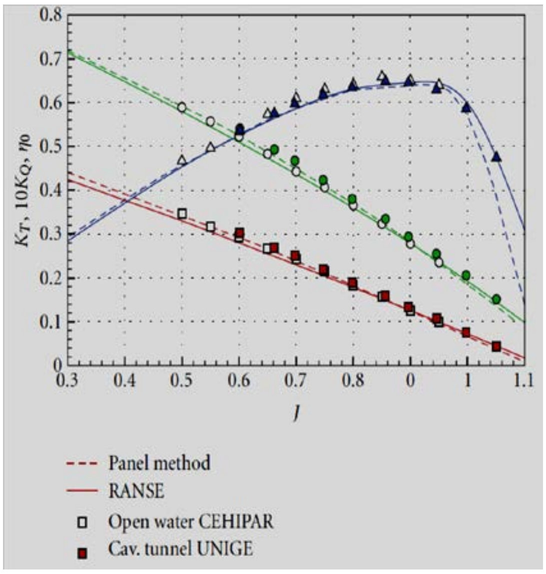


Figure 50: Open water propeller characteristics from RANS / Panel method / Experiments at the model scale (Nielson *et al.*, 2012)

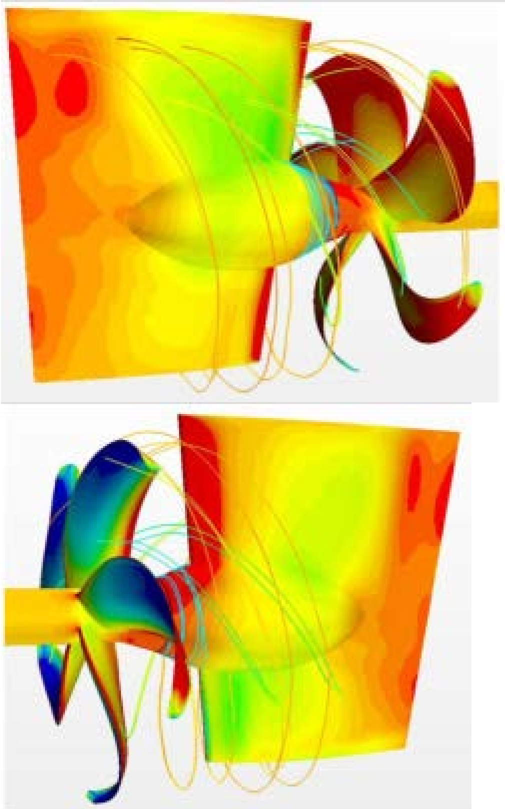


Figure 51: Pressure distribution and streamlines for a KAPPEL propeller with a rudder bulb seen from starboard and port sides
(Nielson *et al.*, 2012)

6.2.2 Propeller Boss Cap Fins (PBCF)

The PBCF is a relatively compact and cheap energy saving device. The effects of which are related to the propeller’s radial loading distribution. If the loading around the hub region is large, the rotational energy of the hub vortex can be effectively recovered by the PBCF.

Ouchi (1989) reported the comparative analyses of the sea trial results of 11 ships and their models. The results indicated considerable scale effects between the model and actual measurements, such that the efficiency gain at full scale could be two to three times that at the model scale. As far as extrapolation methods are concerned, no dedicated procedure for the PBCF has been reported.

Kawamura *et al.* (2012) reported the difference between the effects of PBCF in model tests and in the full-scale data using CFD computations. The efficiency of the computed full-scale value was better than the model test results by around 1%, however it was still almost 2% smaller than the sea trial data.

Hansen *et al.* (2011) reported the analysed results of the efficiency improvement from PBCF installation in more than 60 vessels. Improvements in efficiency of between 2% and 10% were shown, with an average improvement of 5%. Full-scale tests were carried out to find the correlation between the model and full scale results. The full scale results, showing an efficiency gain of around 4%, were a slightly less than the model test results which indicate that no large scale effect is present which is a different conclusion from Ouch (1989). The hull condition was also examined to assess the full-scale results sensitivity.

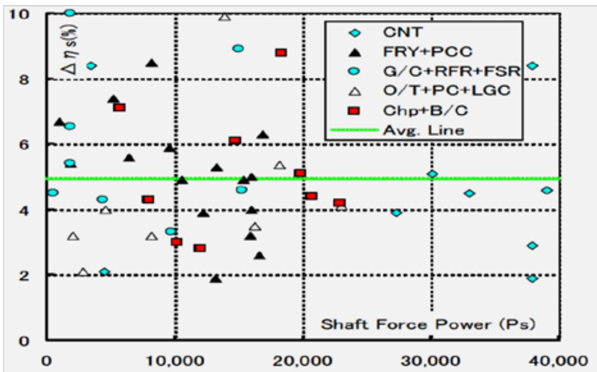


Figure 52: Re Relationship between M/E output and FOC saving using PBCF
(Hansen *et al.*, 2011)

Figure 53: Fitting the PBCF to a full scale ship (Hansen *et al.*, 2011)

Hsin *et al.* (2011) has carried out computations on the unsteady forces in contra-rotating propellers using RANS code and the BEM method. Two CRPs were chosen to be examined and the experimental results were compared. Overall there was a reasonable correspondence between the two, except for the torque of the aft-propeller

6.2.3 Rim driven

Yakovlev *et al.* (2011) compared the rim driven propeller of both the hub and hubless types. The thrust and torque of the hubless propeller are higher than those of the propeller with the hub, whilst the efficiency is almost same. Qing-ming *et al.* (2011) investigated the rim driven propeller (hubless propeller) with four different pitch distributions to examine the performance variations. It was shown that the vortex at the hub is closely related to the radial loading distribution of the propeller.

Cao *et al.* (2012) designed and analysed a rim driven thruster using a CFD code coupled with lifting line theory. The computed results have a good correlation with the experimental data. The computed results also indicated that the correct adjustment of the blade loading distribution can restrain the root and the tip region vortex.

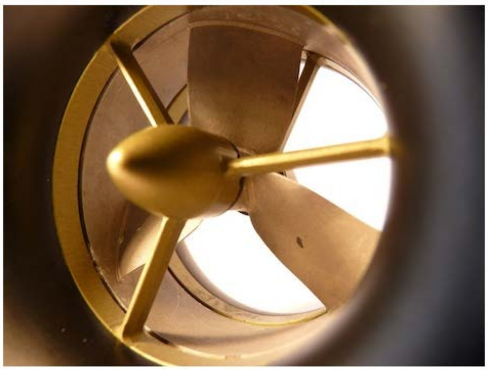


Figure 54: Rim driven propeller (Superyacht News.com, 2011)

6.3 Devices behind the propeller

The most well-known device which can be located behind the propeller may be the Grimse vane wheel, whose efficiency is known to be around 6%. This device however has been used only reluctantly recently as a result of the possibility of damage due to the free running condition and its large diameter.

Figure 55: Grim vane wheel (Source unknown)

Unconventional rudders have recently been the focus for energy savings as well as for the reduction of cavitation problems on the rudder surface. Additional thruster fins (including a post-swirl stator) and a rudder-bulb (including a costa bulb) have been further developed. As the bulb size increases, the efficiency of the propeller can also be higher due to the smaller contraction of the slip stream caused by the rudder bulb. There have been very few studies recently into devices located behind the propeller, which may be due to the complex flow pattern at the stern.

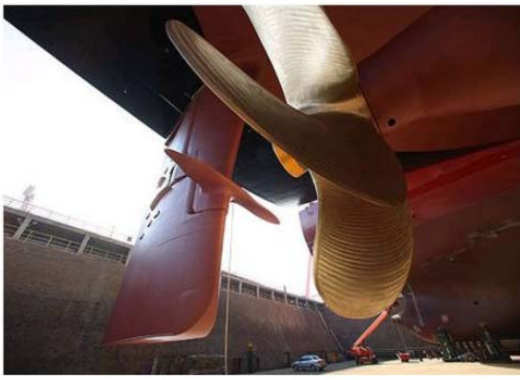


Figure 56: Thrust fin
(Hyundai heavy industry)

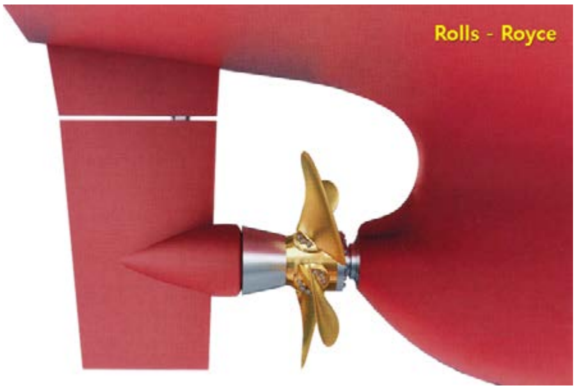


Figure 57: Rudder bulb
(Rolls-Royce brochure)

6.4 Oscillating propulsor.

Mattheijssens *et al.* (2012) reported the analysis of an oscillating foil with a combined motion of heaving and pitching from experiments and a numerical approach. The efficiency of the passive horizontal foil is very high at the design frequency and half of one chord length depth.

Politis and Tsarsitalidis (2012) reported the Flexible Oscillating Duct as a novel propulsor. BEM theory was used for the theoretical analysis of this propulsor and the efficiency gain obtained was between 3% and 10% when compared to a B-series propeller.

7 EXAMINE METHODS OF WAKE SIMULATION

For cavitation tests and measurements of hull pressure fluctuations, the correct simulation of the full-scale wake field is an important technique for the reduction of scale effects. One possible solution is to use a model *that does not* have complete geometrical similarity but is shaped to produce the full-scale wake field. In this case, the full-scale wake field is calculated by the use of CFD tools. Such models are often called “smart dummy” models.

In Germany the joint research project KonKav II, “Correlation of Cavitation Effects Under Consideration of the Wake Field” has been initiated. Participants are Flensburger Schiffbau-Gesellschaft (FSG), Hamburgische Schiffbau-Versuchsanstalt (HSVA), Schiffbau-Versuchsanstalt Potsdam (SVA), Technische Universität Hamburg-Harburg (TUHH) and Universität Rostock (UniHRO). The purpose of the research project KonKav II is to develop a more accurate and marketable cavitation prognosis. In the project the focus is on scale effects that occur through the interaction between wake field, propeller cavitation and the resulting pressure fluctuations. A deeper understanding of these processes will help to convert model test results to full-scale predictions in a reliable way.

One of the objectives is to improve simulation of a full-scale wake field in the context of model tests. It is common practice to use dummy models with attached strainers influencing the flow in a way that the wake field of the full-scale version is simulated. A procedure based on an adjoint sensitivity analysis has been developed by Technical University

Hamburg-Harburg to find appropriate dummy model geometry and appropriate mesh parameters of the strainers to simulate the previously calculated full scale wake field. The photographs (Figure 58) show the geometry of the additional grids used on the conventional dummy model and a dummy model that is optimised on the base of an adjoint sensitivity analysis.

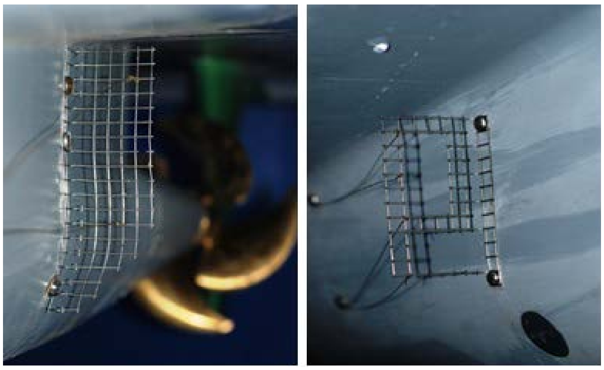


Figure 58: Additional wire grids for the conventional dummy model (left) and the optimised dummy model (right), Photograph from Schiffbau-Versuchsanstalt Potsdam

This research project is on-going, and more results will be available within the next committee.

Schuiling *et al.* 2011, Bosschers *et al.* 2012 and Johannsen *et al.* 2012 report the use of such smart dummy models. Simple shortening and narrowing the model (Figure 59) did not lead to sufficient wake fields. Good results could be achieved by modifying the gondola only in a way that propeller clearance and shaft height was kept the same. The result is shown in Figure 60.

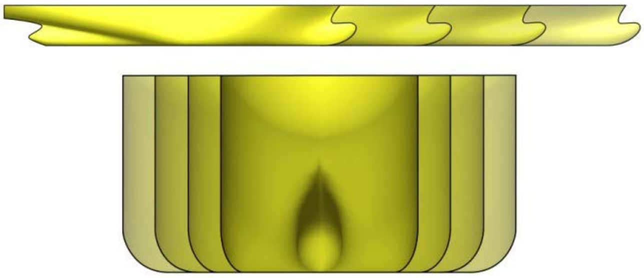


Figure 59: Examples of intermediate forms in systematic hull form variations of width and length.

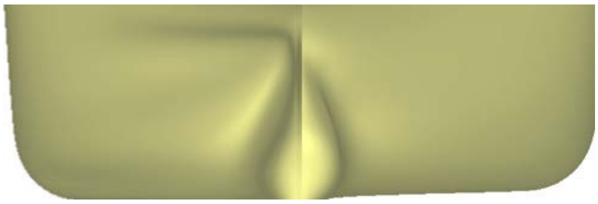


Figure 60: View from behind the Smart Dummy design (left), and the original geosim hull (right)

Even though the full-scale wake field was not fully represented, a good similarity in the upper part was achieved (Figure 61).

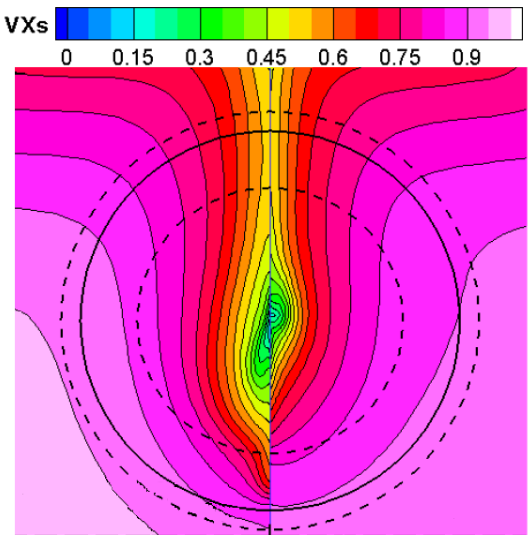


Figure 61: Axial wake velocities of the Smart Dummy design (left) compared to those at ship scale (right). The dashed lines are at 1.1 and 0.6 times propeller radius. The solid line is at the propeller radius

Blade rate order hull pressure fluctuations were close to those found on full scale, but higher order blade rate components were not improved with respect to the full scale. The resulting hull pressure fluctuations for a sensor directly above the propeller were compared. The full- scale measurements were made available by Lloyd's Register. The geosim results were obtained using the wake peak identity approach while the smart dummy results were obtained using thrust-identity. A very good agreement between model scale tests and full-scale tests was obtained for the first harmonic while for the second harmonics there is still a considerable difference.

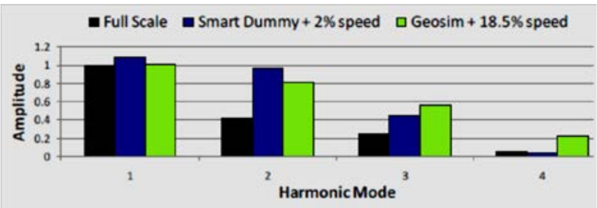


Figure 62: Comparison of non-dimensional full scale (in black) and model scale (in blue and green) pressure amplitudes for first four blade rate orders.

In Heinke, *et al.* (2011), the scale effects on cavitation and fluctuating pressure were experimentally investigated for the cases of a container ship without and with the WED or the VGF fitted. Based on RANS simulations, a shortened dummy model was adopted to simulate the "full-scale wake". In all the cases, the pressure pulses were found to be apparently lower in the "full-scale wake" than in the model wake. Either the WED or the VGF could further reduce the pressure pulses. However, the variations in the fluctuating pressure amplitudes with the attack angle of VGF were found to be different at model- and full-scale, which might provide some hint as to how to optimize the orientation of the VGF.

Further publications deal with adjoint RANS for hull form optimisation, such as Stück *et al.* (2010), Kröger *et al.* (2011), and Rung *et al.* (2012). Various objective functions are considered, among other things the wake quality. Further developments in this field could also be useful for an effective and fast design of smart dummy models.

8 WAKE FRACTION SCALING FOR TWIN SCREW SHIPS

This is related to the 1978 Performance Prediction method in which no wake fraction scaling is given for twin skeg cases.

For cases of finer forms where the shaft is supported by A-brackets using $w_{TS} = w_{TM}$ is still advised and appears to be the general practice.

However, there is an increasing number of fuller form twin skeg vessels. In these vessels the wake field experienced by the propeller is close to that of a single skeg vessel and it might be that the normal single screw wake-scaling procedure should be used. This procedure is used by several establishments.

The use of wake scaling on twin skeg ships needs to be examined further and the next committee should seek to obtain examples of model and ship data so that the issue of wake scaling can be examined. In particular the issue of whether the full wake scaling as used in the single screw method or some reduced level of scaling should be used.

Ohmori *et al.* (2013) have studied the scaling of results from a twin skeg model by CFD and by three semi-empirical techniques. They conclude that the method due to Tanaka gave the best result, but also that the axial and tangential components of wake may need to be scaled separately. They conclude that more work is needed on the scaling of tangential wake.

Sakamoto *et al.* (2011) used CFD backed by model tests to examine wake scaling on twin skeg ships. The work suggests that the ITTC78 single screw scaling method gives reasonably good results, but further work to develop specific twin skeg wake scaling methods is suggested.

9 SCALING OF CONVENTIONAL AND UNCONVENTIONAL PROPELLER OPEN WATER DATA

For initiating a comparative CFD-calculation project two propeller geometries that are in the open domain had to be found. For the conventional propeller the PPTC-propeller (Potsdam Propeller Test Case) from SVA Potsdam could be used. But geometry for an unconventional propeller could not be found.

The PPTC-propeller has 5 blades and a diameter of 250 mm. The following test description was given by the ITTC:

- The propeller shall be tested in a pull configuration. The corresponding hub cap is provided.
- The extent of the shaft behind the propeller has to be at least two propeller diameters.
- The extent of the solution domain can be chosen arbitrarily, however it is considered necessary to have a radial domain extent which gives a cross sectional area of the domain which is at least 100 times larger than the corresponding propeller disc area. Radial extent $D_{domain} > 10\ D_P$
- The dimensionless wall distance on the propeller blade shall be chosen such that the viscous sublayer is resolved.
- It is highly recommended to conduct the calculations in model scale under consideration of laminar-turbulent transition on the blade. For the full-scale calculations a fully turbulent inflow can be assumed.
- The calculations shall be carried out neglecting cavitation.
- The water characteristics shall be taken for a water temperature of T_w = 15°C as provided in Tab. 1 on page 2.1.
- The propeller is a controllable pitch propeller. The gap between hub and blade root is considered unimportant regarding the integral values of the propeller and shall not be taken into account.
- The decision to calculate a single blade passage or the entire propeller (5 blades) is left to the participant.

With respect to the test results the following evaluation was requested by the ITTC:

- Two different scale ratios (λ = 12 and λ = 1).
- Five different advance coefficients: J = 0.6, 0.8, 1.0, 1,2 and 1.4
- The forces on the propeller blade and on blade sections

The thrust and torque of different blade sections shall be evaluated. The coefficients shall be subdivided into a pressure and a frictional component. The intention is to obtain a deeper and more detailed insight into the scale effects of the propeller. The total thrust and torque is obtained via the summation of the different blade section values. The calculations shall be conducted in model and full-scale.

In total 10 calculations were requested. The results are not linked to the corresponding participant, guaranteeing anonymity.

All ITTC members were invited to participate on the comparative CFD-calculation by email. On the SVA Potsdam web site (www.sva-potsdam.de/pptc_ittc_switch.html) the propeller data, an Excel sheet for the results and instructions for the calculations are provided.

In total 9 institutions participated at the ITTC propeller benchmark. Some institutions provided multiple results, giving in total 12 results. The following the institutions participated:

- DGA Hydrodynamics
- Hyundai Heavy Industries
- Krylov State Research Centre
- MARIC
- SJTU
- SSPA
- SSSRI
- SVA
- Technical Research Centre Japan Marine United Corporation (JMU)

The computational results were evaluated for the following:

- Computational method, approach
- Open water curves
- Thrust and torque on blade sections

The participants were asked to fill out a questionnaire and provide details regarding their computations. From this, the following can be drawn:

- In general a single blade passage with periodic side boundaries is used
- The side boundaries are in general matching
- In general unstructured meshes consisting of tetrahedral elements with a prismatic boundary layer and local grid refinment are used
- In model scale the dimensionless wall distances ranges between 1 and approx. 50
- In full-scale the dimensionless wall distances ranges in general between 1 and approx. 30
- The number of cells on the blade surface in the range between 9,800 and 80,000
- For the domain extent two groups can be distinguished. One keeping the domain very large with the cross sectional area of the domain being 3600 times as large as the propeller disc area. The other group has the domain extent very small having values of below 16. The same applies for the upstream and downstream extent of the solution domain.
- All participants use 2 equations turbulence models

The calculated open water curves are compared with the corresponding measurements. The measured model scale open water curves are extrapolated to full-scale according to the ITTC extrapolation method. The extrapolated model data is denoted as EFD (experimental fluid dynamics) results. The CFD (computational fluid dynamics) and EFD results are compared.

The thrust and torque generated by different blade sections, ranging from the hub to the tip of the propeller, are investigated for the requested advance coefficients, both for model and full-scale. The following two diagrams show the K_T and K_Q curves, for the model scale and as recalculated by the ITTC 78 method for ship scale, in comparison with CFD data for full scale. Whilst the K_T correction according to ITTC 78 method is very small, the CFD results show bigger corrections. For KQ the CFD results show positive corrections for higher advance coefficients. The standard deviation of ΔK_T and ΔK_Q is greater than the values itself. That may be caused by the small size of the calculation set. Thus these diagrams are preliminary results. More calculations are necessary.

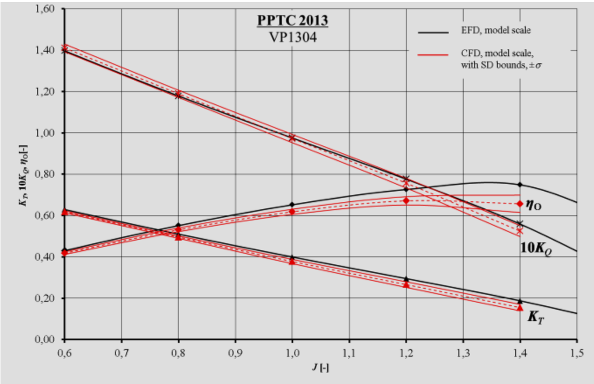


Figure 63: Mean CFD values with σ-bounds and comparison with open water curves in model scale.

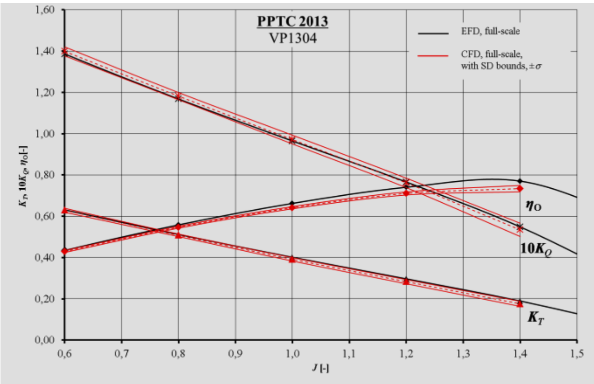


Figure 64: Mean CFD values with σ-bounds and comparison with open water curves in full scale.

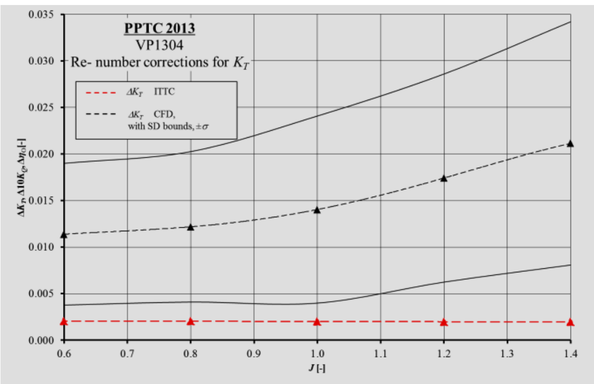


Figure 65: ΔK_T, red acc. to ITTC 78, black from CFD with σ-bounds.

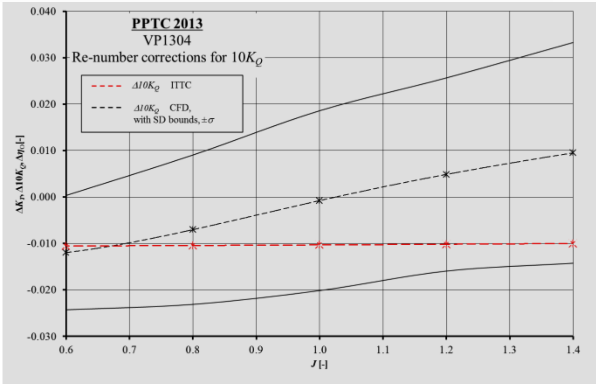


Figure 66: ΔK_Q, red acc. to ITTC 78, black from CFD with σ-bounds

Streckwall, *et al.* 2013 reported their work on an advanced scaling procedure for marine propellers. Emphasis is put on propeller designs with blade shapes that differ from "conventional" type. The work was performed within the European project "PREFUL" with the target to investigate the possibilities of improvements of the scaling calculation in order to consider the differences between blade shapes more precisely. As a result the differences between the several scaling procedures are shown, especially in comparison to the results of a new "strip method", which was developed within the project. It is understood that the enhanced open water corrections are to be compared with full-scale observations (trial trips) in future.

10 DEVELOP GUIDELINES FOR HYBRID PROPULSOR TESTING

10.1 Purpose

Social demand on energy saving and greenhouse gas emission reduction is generating pressure to develop new more efficient propulsors. Remarkable advances in hybrid propulsors (propulsion systems consisting of more than one type of propulsor) has been made in recent years. But the model testing procedure for such propulsors systems has not yet been established.

Among several configurations of hybrid propulsor, the committee focused on the Hybrid Contra-Rotating Shaft Pod propulsor (HCRSP or so-called hybrid CRP concept) and has developed model open water and propulsion test procedure guidelines.

10.2 Definition and Classification

There exist many combinations of hybrid propulsors. They can be classified into two major groups: low interaction group and high interaction group. The high interaction group consists of different propulsors arranged in line in fairly close proximity (e.g. CASE III of Table 1 in Paragraph 3). All other configurations (parallel or in line propulsors with more distance in between) are usually classified into low interaction group (e.g. CASE II and IV of Table 1 in Paragraph 3).

As described in the new guideline, model test of low interaction group can be conducted following the conventional Propulsion Test Procedure 7.5-02-03-01.1 or Podded Propulsor Test Procedure 7.5-02-03-01.3 or Waterjet Propulsion Performance Prediction – Propulsion Test and Extrapolation 7.5-02-05-03.1. However load-varying test should be conducted for each propulsor separately to determine the self-propulsion point.

10.3 Description of guideline

A guideline for the high interaction case was developed. Although many combinations of propulsor (*e.g.* conventional propeller, POD, waterjet, Z-drive, CRP and so on) are possible, effective combinations from the viewpoint of energy efficiency are limited. Thus the guideline focuses on the one of the more significant combinations, HCRSP propulsors. The method is based on the studies by Sasaki (2006/2009), Chang (2011), Quereda (2012), and Sánchez-Caja (2013).

The tank model test consists of propulsor open water test, resistance test and self-propulsion test. The unique point of hybrid propulsor model test is that load distribution between fore propeller and aft POD is not fixed. Thus the load distribution varying tests are compulsory in both propulsor open test and self-propulsion test.

Another point is the arrangement of the propeller open boat for the fore conventional propeller. As for the aft POD drive located behind the fore propeller, the propeller open boat must be arranged in reversed configuration (in front of the fore propeller). In the reversed configuration, viscous wake of the propeller open boat flows into the propeller and the measured results are to be appropriately corrected (Ohmori, 2013).

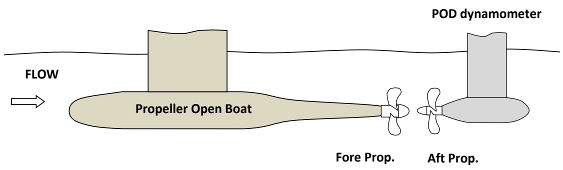


Figure 67: Open water test configuration

The final guideline is registered as 7.5-02-03-01.6 Hybrid Contra-Rotating Shaft Pod Propulsors Model Test.

10.4 Discussion

The procedure covers only model tests and the scaling method is not included. The reason for this is the lack of full-scale validation data. Although the scaling method for podded propulsors will be helpful, the development of the full-scale prediction method is the subject of future work.

Mitsubishi Heavy Industries (MHI) is the first shipyard that has built a large ship with HCRSP (Ueda, 2004). Their tank test procedure is simpler than the new guideline (it is basically based on the conventional propulsor test procedure), but it has already been confirmed by the full scale trial results. However the committee could not adopt this method as no published paper is available.

Figure 68: HCRSP mounted on a ROPAX Ferry (Ueda, 2004)

11 MONITORING OF FULL SCALE DATA FOR PODDED PROPULSION.

No new full-scale data has been published, the only known available example is the "ABB case". The Propulsion committee contacted ABB to establish if such data could be published.

The discussion in the 25th ITTC POD committee report concluded that the main issue is still the scaling of pod housing drag. Taking this into account, the most helpful results for benchmarking would be full-scale data with measurements of propeller thrust, torque and also pod housing drag.

The propulsion committee, with reference to Honninen, *et al.* (2007), discussed with ABB if such full-scale data could be made available for the ship "Norilsky Nickel" for which extensive full-scale measurements were conducted, including simultaneous measurements of the loads on the thruster body and propeller. ABB express its willingness to acquire and analyze such data and to publish the results in future.

12 CONCLUSIONS

12.1 Recommendations to the Next Committee

12.1.1 Procedure Review/Update

The 27th Propulsion committee has developed a new guideline for HCRSP (Hybrid Contra-Rotating Shaft Pod) Propulsor Model Tests. The model test scaling is not discussed in this guideline as there is a lack of model test and full-scale trials comparison data. For this reason, the committee recommends the continued monitoring of model test and scaling procedures used for this kind of device (and for propulsion devices in general) by member institutions of the ITTC. If there is sufficient information on the comparison between model and full scale data indicating changes to test or scaling procedures, the relevant guidelines should be updated.

The committee recommends that the monitoring of the existing literature for examples of the Reynolds number scale effects should continue in order to update the 7.5-02 -05-03.2 Waterjet System Performance procedure. Clarification and detailing of the procedure in the part relating to the data acquisition and in the part related to extrapolation is required. Further reviews of the literature should examine the need and use of the blade-tip and chord Reynolds numbers as well the intake duct Reynolds number and update the procedure if required.

The committee recommends a review of the power margins given in the guidelines and the monitoring of the effect of the introduction of the EEDI on power margins.

12.1.2 New Procedures

The committee recommends developing a new procedure for propulsion performance prediction for triple shaft vessels. Although the existing procedure largely covers this, the procedure needs to be extended to take account of the interaction between centerline and wing propulsors and to allow for the determination of the wake fraction and the thrust deduction factors for these propulsors separately.

12.1.3 Technologies to Monitor

The committee recommends monitoring the model test and scaling procedures for energy saving device technologies. The impact on self-propulsion and cavitation testing is also to be reported to assess the way of taking into account differences in inflow speed and direction between the model and the ship. This is particularly relevant to wake improving devices where the optimum alignment may be different between the model and ship. This raises the question of the relevance of any propulsive gains determined from model tests to the ship. The use of CFD and/or a combination of CFD and EFD should be also considered, as well as full-scale trials results.

The committee recommends monitoring the "smart dummy" model used for cavitation tests for propulsors behind a skeg. A joint research project "Koncav II" has been launched in Germany and preliminary results might be available within the three years term of the next committee.

The committee recommends examining the existing procedures and assessing where CFD results can be introduced in the propulsion process to assist EFD e.g. use CFD to determine the target wake field to be modelled in cavitation testing, use of smart dummy model to model the target wake field, use of CFD in conjunction with EFD for composite propellers).

The present committee was not able to find a suitable model test and CFD study for unconventional propellers. The definition of what would be interesting to work with as an unconventional propulsor is still to be discussed with the CFD committee.

Recent publications suggest that RANS codes are more and more widely used for propulsor design. The analysis of the data of the benchmark launched by the 27th committee should be able to give some answers to the use and interpretation of RANS methods and procedures. A combine effort with the CFD committee will encourage the continuation of this study with the aim of getting further contributions from member institutes. The EFD data used in this study comes from only one institute. The committee recommends that additional EFD studies on the same propeller design should be performed.

An area to examine is the fluctuating components of propeller bearing forces, especially on Pod and azimuthing thruster

Further work is still required on the way to test and analyse the results for composite propellers. The use of CFD in combination with EFD to investigate the fluid-structure interaction (static and dynamic hydro-elastic response) needs to be better understood.

Experimental techniques such as detailed local flow velocity measurement using PIV, blade strain, cavity surface and volume measurement still need to be monitored.

Testing and estimation methods for propulsors in bubbly flow should be monitored. The open-water and self-propulsion characteristics in bubbly flow are relevant to air lubricated vessels and the effect of void fraction, void type and affected area on the propeller need to be investigated and understood.

12.1.4 Scaling for Propulsors

As no full scale data on Pod Propulsor have been found within this session, the committee recommends that the next committee should continue to look for-full scale data.

The prediction of full scale cavitation induced hull pressure is still of interest either by means of CFD directly or CFD and EFD used in combination.

In view of the growing interest in energy saving devices further work is still required on model test techniques and the prediction of power savings.

12.2 Recommendations to the Conference

The Committee recommends to the Full Conference that they should

- Adopt the revised procedure ITTC Procedure 7.5-02 03-01.4 1978 ITTC Performance Prediction Method
- Adopt the revised procedure ITTC Procedure 7.5-02 03-02.3 Propulsor Nominal Wake Measurement by LDV Model Scale Experiments
- Adopt the revised procedure ITTC Procedure 7.5-02 03-03.2 Testing and extrapolation Methods Propulsion : Cavitation Description of Cavitation Appearances
- Adopt the revised procedure Update to ITTC Procedure 7.5-02 03-03.3 Cavitation Induced Pressure Fluctuations Model Scale experiments
- Adopt the revised procedure ITTC Procedure 7.5-02-03-03.4 Cavitation Induced Pressure Fluctuations: Numerical Prediction Methods
- Adopt the new guideline 7.5-02-03-01.6 HCRSP (Hybrid Contra-Rotating Shaft Pod) Propulsors Model Test

REFERENCES

Amini H., Sileo L., Steen S., 2012, "Numerical Calculations of Propeller Shaft Loads on Azimuth Propulsors in Oblique Inflow", Journal of Marine Science and Technology, 17(4): 403-421.

Amini H. and Steen S., 2011, "Experimental and Theoretical Analysis of Propeller Shaft Loads in Oblique Inflow", Journal of Ship Research, Vol. 55, No. 4, pp. 268-288.

Barkmann U., Heinke H.J., Lübke L., "Potsdam Propeller Test Case (PPTC), Test Case Description", Second International Symposium on Marine Propulsors smp'11, Hamburg, Germany, June 2011, Workshop: Propeller performance

Black S. , "Composite propeller for Royal Navy minehunter", http://compositesworld.com/articles/, 2011.

Bertetta, D., Brizzolara, S., Canepa, E., Gaggero, S and Viviani, M., 2012, "EFD and CFD characterization of a CLT propeller", International Journal of Rotating Machinery, Article ID 348939.

Bono,B., Cucinotta,F.,Guglielmino,E.,Zotti,I., 2012, "Experimental and Numerical Evaluation of an Innovative Planning Hull", NAV 2012

Bosschers J. and van Wijngaarden E., "Scale effects on hull pressure fluctuations due to cavitating propellers", 10th International Conference on Hydrodynamics October 1-4, 2012 St. Petersburg, Russia

Bosschers J., Lafeber F.H., de Boer J., Bosman R. & Bouvy A., "Underwater radiated noise measurements with a silent towing carriage in the Depressurized Wave Basin", AMT '13 17-18 September 2013, Gdansk, Poland.

Briançon L., Fournier P. & Frechou D., "Marine propeller noise measurements techniques in hydroacoustics tunnel", AMT '13 17-18 September 2013, Gdansk, Poland.

Brouwer J., Hagesteijn G., Bosman R., "Propeller-ice impacts measurements with a six-component blade load sensor", 2013, Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Bulten, N., Nijland, M., 2011, "On the Development of a Full-Scale Numerical Towing Tank Reynolds Scaling Effects on Ducted Propellers and Wakefields", Second International Symposium on Marine Propulsors 2011, Hamburg, Germany.

Bulten N., Suijkerbuijk R., 2013, "Full Scale Thruster Performance and Load Determination Based on Numerical Simulations", Third International Symposium on Marine Propulsors (SMP'13), Launceston, Australia.

Cao, Q.-M., Hong F.-W., Tang, D.-H., HU, F.-L., Lu, L.-Z., 2012, "Prediction of Loading Distribution and Hydrodynamic Measurements for Propeller Blades in a Rim Driven Thruster", Journal of Hydrodynamics, Vol. 24, Issue 1, pp. 50-57.

Cao, Q.-M., Dong, A., Hong, F. and Tang, D., 2012 "Numerical Design of Rim Driven Thruster and Hydrodynamic Measurement, International Marine Design Conference 2012, Vol 3, pp. 479-489.

Carlton, J. S., 2012, "Marine propellers and propulsion", Butterworth Heinemann, Great Britain.

Castro A. M., Carrica P. M., Stern F., 2011, "Full scale self-propulsion computations using discretized propeller for the KRISO container ship KCS", Computers & Fluids, 51, pp.35~47.

Chang, S.-H., Kinnas, A., 2012, "Numerical Simulation of Wetted and Cavitating Flows inside Water-jets", 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden

Chang, B-J., Go, S., 2011, "Study on a procedure for propulsive performance prediction for CRP-POD systems", J. Marine Sci. Technology, Vol. 16 pp.1-7.

Cheng, H., Chien, Y., Hsin, C., Chang, K., Chen, P., 2010, "A numerical comparison of end-plate effect propellers and conventional propellers", 9th International Conference on Hydrodynamics 2010, Shanghai, China, pp.495-500

Dang J., Liu R., Pouw C.,2012,"Waterjet System Performance and Cavitation Test Procedures", Symposium on Marine Propulsors SMP'13,Launceston,Australia.

Dang J., Brouwer J., Bosman R. and Pouw C., 2012, "Quasi-Steady Two-Quadrant Open Water Tests for the Wageningen Propeller C- and D-Series", Twenty-Ninth Symposium on Naval Hydrodynamics, Gothenburg, Sweden.

Dang,J., van den Boom, H.J.J., Ligtelijn, J.Th.,2013, The Wageningen C- and D-Series propellers, FAST '13

Delaney, K.,2011, "Use of RANS for analysis of a high speed Sealift Concept Vessel, MARINE 2011,Lisbon,Portugal

DNV, 2010, "Rules for Classification Of Ships, Silent Class Notation", Part 6, Chapter 24. Det Norske Veritas.

Dubbioso G., Muscari R., Di Mascio A., 2013, "CFD Analysis of Propeller Performance in Oblique Flow", Third International Symposium on Marine Propulsors (SMP'13), Launceston, Australia.

Epps.Dual Operating point Blade optimization for high speed propellers. FAST 2011

Eslamdoost, A., Larsson, L., Bensow, R.,2013, "Contribution of Different Parameters to the Thrust Deduction of a Waterjet Propelled Hull", FAST 2013

Ferrieri J.M., Veitch B., and Akinturk A.,"Experimental Study on Ice Management through the use of Podded Propeller Wash", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Fujisawa T., 2013, "Study of Scale Effect on CRP by viscous CFD", Conference Proc. JASNAOE, 17: 147-148.

Funeno I., Pouw C., Bosman R, "Measurements and Computations for Blade Spindle Torque of Controllable Pitch Propellers in Open Water", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Fu. Measurements and Computational Predictions of a Deep-V MonohullPlaning Hull FAST 2011

Gany, A., Shemer,A., Gofer,A., Har-Lev,D., 2008, "Testing Air-Augmented Waterjet Propulsion", International Conference on Waterjet Propulsion, London, UK

Giles W., Dinham-Peren T., Amaratunga S., Vrijdag A., Partridge R., "The Advanced WaterJet: Propulsor Performance and Effect on Ship Design", 10th IMarEST Naval Engineering and Exhibition , May 2010, Portsmouth, UK

Głodowski R., Richards J., Koushan K., Alterskjaer S.A., Atlar M. & Saisto I., "Open water performance characteristics of a pod propulsor– the benchmark test", The 3rd International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, AMT'13, Gdansk, Poland.

Gowing S., 2011, "Experimental Study of an Air-Augmented Waterjetpropulsor", Symposium on Marine Propulsors'11, Hamburg, Germany

Guiard, T., Leonard, S. and Mewis F., "The Becker Mewis Duct-Challenges in Full-Scale Design and new Developments for Fast Ships", 2013, Third International Symposium on Marine Propulsors SMP '13, Tasmania, Australia.

Haimov H., Vicario J., Del Corral, J., 2011, "RANSE Code Application for Ducted and Endplate Propellers in Open Water", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Hansen, H. R., Dinham-Peren, T., Nojiri, T., 2011, "Model and Full Scale Evaluation of a 'Propeller Boss Cap Fins'Device Fitted to an Aframax Tanker", Second International Symposium on Marine Propulsors 2011, Hamburg, Germany.

Heinke H.-J., Hellwig-Rieck K., 2011, "Investigation of Scale Effects on Ships with a Wake Equalizing Duct or with Vortex Generator Fins", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

He X.D., Hong Y., Wang R.G., "Hydroelastic optimization of a composite marine propeller in a non-uniform wake", Ocean Engineering Journal 39 (2012) 14–23.

Henderson K, "New Design of 2000 TEU Feeder", http://articles.maritimepropulsion.com/article/New-Design-of-2000-TEU-Feeder-6784.aspx, Maritime propulsion, January 2012.

Hideki Y., Hiroaki M., Aiichiro Saeki, 2011, "Energy Saving Technology of the Diesel-Electric Propulsion, System for Japanese Coastal Vessels", IHI Engineering Review, Vol 44

Himei K. Numerical Analysis of Unsteady Open Water Characteristics of Surface piercing Propeller. Third International Symposium on Marine Propulsors SMP'13, Launceston,Australia

Hollenbach, U. and Reinholz, O., 2011, "Hydrodynamics Trends in Optimization Propulsion", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany

Holtrop J., 2000, "MARIN Training Course on Propulsion Hydrodynamics", Netherlands.

Huang S.-F., Huang G.-F., 2012, "Numerical Investigation for Effectiveness of Wake Equalizing Duct", 10th International Conference on Hydrodynamics, St. Petersburg, Russia.

Huang, S., Huang, Z., Branch, S., 2012," Investigation on Energy-Saving Effectiveness of Pre-Swirl Duct Based on CFD", 10th International Conference on Hydrodynamics, St. Petersburg, Russia

Hur J. W., Lee H., Chang B. J., 2011, "Propeller Loads of Large Commercial Vessels at Crash Stop", Second International Symposium on Marine Propulsors (SMP'11), Hamburg, Germany.

Hwang, J-L.,Chin, S-S.,Chang,K-K., Tu, Chin-Chin.,2011, "Development of the Trans-Velocity Propellers", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany

Inukai, Y., 2011, "Development of Contra-Rotating Propeller with Tip-Raked Fins", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Inukai, Y., "A Development of a Propeller with Backward Tip Rake Fin", 2013, Third International Symposium on Marine Propulsion SMP'13, Tasmania, Australia.

Ishida, S., Fujino, R., 1998, "A Performance Prediction Method for Contra-rotating Propeller Ships", Society of Naval Architects of Japan, Vol. 183, pp. 35-44, 1998 (in Japanese).

Jang H., Mahesh, K., 2012, "Large Eddy Simulation of Crashback in Ducted Propellers with Stator Blades", 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden.

Jang H., Mahesh, K., 2013, "Large Eddy Simulation of Flow around a Reverse Rotating Propeller", Journal of Fluid Mechanics, 729: 151-179.

Johannsen, C., Wijngaarden, E., 2012, "Investigation of Hull Pressure Pulses, Making Use of Two Large Scale Cavitation Test Facilities", Proceedings of the Eighth International Symposium on Cavitation (CAV 2012)

Jang, Jin-Yul, 2013, "Study on Performance of Contra-Rotating Propeller", 2013, Master Thesis, Pusan National University, Korea.

Kandasamy. Optimization of Waterjet Propelled High Speed Ships – JHSS and Delft Catamaran. FAST 2011

Kane C., http://www.compositecarbonfiberprop.com/qinetiq.htm, "World's Largest Composite Carbon Fiber Propeller", 2001

Kawakita C., "Study on Marine Propeller Running in Bubbly Flow", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Kawakita, C., Takano, S., Kodan, Y., & Mizokami, S., (2011) 'Experimental Investigation of the Behavior of Injected Air on the Ship Bottom and its Influence on Propeller'. Journal of the Japan Society of Naval, Architects and Ocean Engineers, Vol.12, pp.43-50.

Kawamura, T., Ouchi, K., Nojiri, T., 2012, "Model and full scale CFD analysis of propeller boss cap fins (PBCF)", Journal of Marine Science and Technology, 0948-4280.

Kawamura T., Ouchi K., Takeuchi S., 2013, "Model and full scale CFD analysis of propeller boss cap fins (PBCF)", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Kim K.-S., Kim Y.-C., Kim J., Lee Y.-Y., Ahn H., Yim G., Park I.-R., 2012, "A Study on the Energy Saving Fin Attached on the Ship Stern", 10th International Conference on Hydrodynamics, St. Petersburg, Russia.

Kim, S.-E., 2009, "A Numerical Study of Unsteady Cavitation on a Hydrofoil", Proc., 7th International. Symposium on Cavitation CAV2009, Ann Arbor, Michigan, USA.

Kinnas S.A., Jeon C.-H., Purohit J., TianY., 2013, "Prediction of the Unsteady Cavitating Performance of Ducted Propellers Subject to an Inclined Inflow", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Korkut, E., 2006, "A Case Study for the Effect of a Flow Improvement Device (a Wake Equalizing Duct) on Ship Powering Characteristics", Ocean Engineering, Vol.33, pp.205-218

Koushan K., Spence S., Savio L., 2011, "Ventilated Propeller Blade Loadings and Spindle Moment of a Thruster in Calm Water and Waves", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Krasilnikov V. I., 2013, "Self-Propulsion RANS Computations with a Single-Screw Container Ship", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Li, D-Q, Grekula,M., Lindell, P. and Hallander., J. (2012). Prediction of Cavitation for the INSEAN Propeller E779A Operating in Uniform Flow and Non-Uniform Wakes. Proceedings of 8th International Symposium on Cavitation, CAV2012, Singapore.

Lu, N. Svennberg,N., Bark, G. and Bensow, R. (2012) Numerical Simulations of the Cavitating Flow on a Marine Propeller. Proceedings of 8th International Symposium on Cavitation, CAV2012, Singapore.

Maciel P., Koop A., Vaz G., 2013, "Modelling Thruster-Hull Interaction with CFD", OMAE ASME 32nd International Conference on Ocean, Offshore and Arctic Engineering, Nantes, France.

Manudha T. Herath, B. Gangadhara Prusty, G. H. Yeoh, M. Chowdhury and Nigel St.John, "Development of a shape-adaptive composite propeller using bend-twist coupling characteristics of composites", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Mattheijssens, J., Marcel, J.-P., Bosschaerts, W., Lefeber, D., 2012, "Oscillating foils for ship propulsion", 9th National Congress on Theoretical and Applied Mechanics, Brussels, 9-10-11.

McDonald. A Tool for Analyzing and Comparing Tri-SWACH, Monohull and Trimaran Hullforms. FAST 2011

Mewis, F., Guiard, T., 2011, "Mewis Duct® - New Developments, Solutions and Conclusions", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Mosaad M.A., Gaafary M.M., El-Kilani H.S., Amin I.A., Power prediction of WIG craft. International Shipbuilding Progress 58, pp219-238, 2012.

Motley M.R. and Young Y.L., "Scaling of the Transient Hydroelastic Response and Failure Mechanisms of Self-Adaptive Composite Marine Propellers", International Journal of Rotating Machinery Volume 2012.

Nielsen, J. R., Shin, K. W., Lundgren, E., Faghani, F., 2012, "Combined Kappel propeller and rudder bulb system for improved propulsion efficiency", MAN Diesel & Turbo, Frederikshavn, Denmark

Ohmori, T., Ochi, F. and Nagaya, S., 2013, "A Study on the Influence of Propeller Open Boat in Reversed Configuration Propeller Open Water Test", Conference Proc. Japan Society of Naval Architects and Ocean Engineers, Vol. 17, Nov. 2013, Japan.

Ohmori T., Sudo Y., "Numerical and Experimental Study on Wake Scaling of Twin-Skeg Ships", The Japan Society of Naval Architects and Ocean Engineers. 2013

Ouchi, K, 1989, "Research and Development of PBCF –Improvement of Flow from Propeller Boss", The Proceedings of the International Symposium on Ship Resistance and Powering Performance '89 (ISRP), Shanghai.

Ouchi K., Uzawa K, Kanai A. and Katori M., "Wind Challenger - the Next Generation Hybrid Sailing Vessel", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Palm M., Jürgens D., Bendl D., "Numerical and Experimental Study on Ventilation for Azimuth Thrusters and Cycloidal Propellers", Second International Symposium on Marine Propulsors ,SMP'11, Hamburg, Germany, June 2011 ,

Park H.-G., Choi J.-K., Kim H.-T., 2013, "An Estimation Method of Full Scale Performance for Pulling Type Podded Propellers", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Pecoraro A., Di Felice F., Felli M., Salvatore F., Viviani M., Propeller-hull interaction in a single-screw vessel, Third International Symposium on Marine Propulsors SMP'13, Launceston, Tasmania, Australia, May 2013

Politis, G., Tsarsitalidis, V., 2012, "Flexible Oscillating Duct: An approach to a novel propulsor", Applied Ocean Research, Vol. 36, 99.36-40.

Pospiech P., "VOITH's New Propulsion System: The Voith Linear Jet (VLJ)", http://articles.maritimepropulsion.com/, November 23, 2012

Poul, A., 1997, "A Comparative Study of Conventional and Tip-Fin Propeller Performance", Twenty-First Symposium on Naval Hydrodynamics 1996, Trondheim, Norway, pp.930-945.

Quereda, R., Veikonheimo, T., Pėrez-Sobrino, M., Ponce, J., Sánchez-Caja, A., Masip, J., González-Adalid, J., Uriarte, A., Nijland, M. and Kokkia, K., 2012, "Model Testing and scaling for CRP POD", Proc. ICHD10, Oct. 2012, St. Petersburg, Russia.

Rijpkema D., Starke B., Bosschers J., 2013, "Numerical Simulation of Propeller-Hull Interaction and Determination of the Effective Wake Field Using a Hybrid RANS-BEM Approach", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Rung T., Wokner K., Manzke M., Brunswig J., Ulrich C., Stuk A., "Challenges and Perspectives for Maritime CFD Applications", Jahrbuch der Schibautechnischen Gesellschaft, 103:127-139, 2009.

Sakamoto N, Kawanami Y, Uto S., “Computational Study for Single-Screw and Twin-Skeg Container Ships in Full Scale by Unstructured Grid based RANS Solver”, The Japan Society of Naval Architects and Ocean Engineers, 2011

Sakamoto N., Kawanami Y., Uto S., Sasaki N., 2013, “Estimation of Resistance and Self-Propulsion Characteristics for Low L/B Twin-Skeg Container Ship by a High-Fidelity RANS Solver”, Journal of Ship Research, 57(1).

Sampson R., Atlar M., St John J.W. & Sasaki N., “Podded propeller ice interaction in a cavitation tunnel”, Third International Symposium on Marine Propulsors SMP’13, Launceston, Australia May 2013.

Sampson R. & Atlar M., 2013, “Prediction of the effect of cavitation on propeller ice interaction”, The 3rd International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, AMT’13, Gdansk, Poland.

Sanchez-Caja, A., Gonzalez-Adalid, J., Perez-Sobrino, M. and Saito, I., 2012, “Study of End-Plate Shape Variations for Tip Loaded Propellers Using a RANSE Solver”, 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden.

Sánchez-Caja A., González-Adalid J., Pérez-Sobrino M., Saisto I., 2014a, “Evaluation of Endplate Impact on Tip Loaded Propeller Performance Using a RANSE Solver”, International Shipbuilding Progress, 61: 103~128.

Sánchez-Caja A., Martio J., Saisto I., Siikonen T., 2014b, “On the Enhancement of Coupling Potential Flow Models to RANS Solvers for the Prediction of Propeller Effective Wakes”, Journal of Marine Science and Technology, published online in February 2014.

Sánchez-Caja A., González-Adalid J., Pérez-Sobrino M., Sipilä T., 2014c, “Scale Effects on Tip Loaded Propeller Performance Using a RANSE solver”, Ocean Engineering, http://dx.doi.org/10.1016/j.oceaneng.2014.04.029.

Sánchez-Caja A, Pérez-Sobrino M., R. Quereda M.R., Nijland N., Veikonheimo T., González-Adalid J., Saisto I. and Auriarte A., “Combination of Pod, CLT and CRP Propulsion for Improving Ship Efficiency: the TRIPOD project”, A Third International Symposium on Marine Propulsors SMP’13, Launceston, Australia May 2013.

Sasaki, N., Kawanami, Y., Ukon, Y., Kano, T. and Tomizawa, S., 2006, “Model test procedure and analysis of hybrid CRP POD system”, Proc. 2nd International conference on technical advances in podded propulsion (T-POD), Nantes, France.

Sasaki, N., Kuroda, M., Fujisawa, J., Imoto, T. and Sato, M., 2009, “On the Model Tests and Design Method of Hybrid CRP Podded Propulsion System of a Feeder Container Ship”, Proc. The First International Symposium on Marine Propulsors SMP’09, June 2009, Trondheim, Norway.

Scherer., “Hydrodynamics of surface piercing outboard and sterndrive propulsion systems”. FAST Conference 2011

Schuiling B., Lafeber F. H., van der Ploeg A., van Wijngaarden E., "The Influence of the Wake Scale Effect on the Prediction of Hull Pressures due to Cavitating Propellers", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany, June 2011.

Schulze, "An improved Z-drive with Contra-Rotating Propellers for High-Speed Applications", FAST Conference 2011

Sileo L., Steen S., 2011, "Lateral Force and Turning Moment on a Reversing Ship", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Sipilä, T. and Siikonen, T. (2012). RANS Predictions of a Cavitating Tip Vortex. Proceedings of 8th International Symposium on Cavitation, CAV2012, Singapore.

Skejic,R.,2012,"Effective Power Requirements for High Speed Catamarans in Calm Water During the Preliminary Design Stage", NAV 2012 - 17th International Conference on Ships and Shipping Research.

Streckwall H, Greitsch L., Scharf M., "An advanced Scaling Procedure for Marine Propellers", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Stuck A., Kroger J., Rung T., "Adjoint-ased Hull Desogn for wake optimization", Ship Technology Research, Vol. 58, No.1, January 2011.

Taketani T., Kimura K., Ando S., Yamamoto K., "Study on Performance of a Ship Propeller Using a Composite Material", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia May 2013.

Tomohiro Takai, MamvannanKandasamy, Frederick Stern, Verification and validation study of URANS simulations for an axial waterjet propelled large high-speed ship, J Mar Sci. Tech. (2011) 16: 434-447

Tsai,J.,Tse,C.,Wang,S.,Yeh,D., 2005, "Experimental Study on the Hydrodynamic Performace of Two-Phase Nozzle for Waterjet propulsion System", FAST 2005, St-Petersburg, Russia.

Ueda, N., Oshima, A., Unseki, T., Fujita, S., Takeda, S. and Kitamura, T., 2004, "The First Hybrid CRP-POD Driven Fast ROPAX Ferry in the World", Mitsubishi Heavy Industries, Ltd. Technical Review Vol.41 No.6.

Van, S.-H., Kim, M.-C., Lee, J.-T., 1993, "Comparison of Powering Performance Prediction Methods for Preswirl Stator Propulsion System", Proceedings of SNAK, Spring Meeting.

Veikonheimo T., 2006, "Solving problems of propeller model scaling", The naval Architect, pp 51-53.

Verma A., Jang H., Mahesh K., 2012, "The Effect of an Upstream Hull on a Propeller in Reverse Rotation", Journal of Fluid Mechanics, 704: 61-88.

Verhulst M. A., 2012, "Statistical Analysis of Model-Full Scale Correlation for Propulsion Tests and the Role of CFD in Extrapolation", 10th International Conference on Hydrodynamics, St. Petersburg, Russia.

Vesting F., Johansson R., Bensow R. E., 2013, "Parameter Influence Analysis in Propeller Optimisation", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Villa D., Gaggero S., Brizzolara, S., 2012, "Ship Self Propulsion with Different CFD Methods: From Actuator Disk to Viscous Inviscid Unsteady Coupled Solvers", 10th International Conference on Hydrodynamics, St. Petersburg, Russia.

Weber, "Application of the New FORTJES Z-drive from REINTJES on Planing Vessels", FAST 2011

Wu, X., S., Choi, J-K., Hsiao, C-T., Chachine G.L., 2010, "Bubble Augmented Waterjet Propulsion: Numerical and Experimental Studies", 28th Symposium on Naval Hydrodynamics, Pasadena, California, USA

Wu, X., Singh, S., Choi, J-K., Chachine G.L., 2012, "Waterjet Thrust Augmentation using High Void Fraction Air Injection", 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden

Xia L., Lundberg J., Bensow R. E., 2012, "Performance Prediction of a Nozzle Propeller", 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden.

Yakovlev, A. Y., Sokolov, M. A., Marinich, N. V., 2011, "Numerical Design and Experimental Verification of a RIM-Driven Thruster", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Yasuhiko, I, Tadaaki, K., Shigeki, N., Fumitoshi, O., 2011, "Energy-Saving Principle of the IHIMU Semicircular Duct and Its Application to the Flow Field Around Full Scale Ships", IHI Engineering Review, Vol. 44, No. 1, pp.17-22.

Young Y.L., "Hydroelastic behaviour of flexible composite propellers in wake inflow", 16th International Conference on Composite material, 2007

Young Y.L.,"Dynamic hydroelastic scaling of self-adaptive composite marine rotors", Elsevier Ltd., Composite Structures Journal 92, pp 97–106, 2010.

Young Y. L, 2012, "Analysis & Scaling of Flexible Composite Marine", Presentation at the 3rd Propulsion Committee meeting, Pusan.

Young Y.L., Motley M.R., 2011, "Influence of Material and Loading Uncertainties on the Hydroelastic Performance of Advanced Material Propellers ", Second International Symposium on Marine Propulsors SMP'11, Hamburg, Germany.

Yu H., Kong W., Cai R., Wang J., Zhang Y., 2013, "Numerical Study of Energy-saving Mechanism of Duct on a VLCC with Real-geometry Propeller", Third International Symposium on Marine Propulsors SMP'13, Launceston, Australia.

Zhang N., Chen X., Xie H., 2012, "Numerical Simulation of Hull/Propeller Interaction of Submarine", 10th International Conference on Hydrodynamics, St. Petersburg, Russia.

Zheng,Y.,Dong,W.,2012,"Hull Form Design and Preliminary Evaluation of a Surface Planing Submersible Ship", ISOPE-2012, Rhodez, Greece

Zondervan, G.-J., Holtrop, J., Windt, J., Terwisga, T., 2011, “On the Design and Analysis of Pre-Swirl Stators for Single and Twin Screw Ships”, Second International Symposium on Marine Propulsors SMP’2011, Hamburg, Germany

Zurcher K., Bose N.,Binns J.R., Thomas G.,Davidson G. Design and commissioning tests for waterjet self-propulsion testing of a medium speed Catamaran ferry using a single Demihull, Third International Symposium on Marine Propulsors SMP’13, Launceston, Australia.