The Propulsion Committee

Final Report and Recommendations to the 26th ITTC

1. INTRODUCTION

1.1 Membership and Meetings

The members of the Propulsion Committee of the 26th International Towing Tank Conference are as follows:

- Dr. Suak-Ho Van (Chairman), Maritime and Ocean Engineering Research Institute, (MOERI), Korea
- Dr. Scott D. Black (Secretary), Naval Surface Warfare Center (David Taylor), U.S.A.
- Professor Jun Ando, Kyushu University, Japan
- Valery O. Borusevich, Krylov Shipbuilding Research Institute, Russia
- Professor Emin Korkut, Technical University of Istanbul, Turkey
- Dr. Anton Minchev, FORCE Technology, Denmark
- Dr. Didier Fréchou, DGA Hydrodynamics, (Bassin d'essais des carenes) France
- Rainer Grabert, Schiffbau-Versuchsanstalt Potsdam GmbH (SVA) Germany
- Professor Chen-Jun Yang, Shanghai Jiao Tong University, China

Four Committee meetings were held as follows:

- MOERI, Korea, 11-13 March 2009
- DGA, France, 4-6 November 2009
- NSWC, USA, 5-7 May 2010,
- SVA, Germany, 9-11 February, 2011

1.2 Recommendations of the 25th ITTC

The 25th ITTC recommended the following works for the 26th ITTC Propulsion Committee:

1. Update the state-of-the-art for predicting for propulsion systems emphasizing developments since the 2008 ITTC Conference. The committee report should include sections on:
   (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 modelling and full-scale measurements.
2. Review ITTC Recommended Procedures relevant to propulsion (including procedures for uncertainty analysis).
   (a) Identify any requirements for changes in the light of current practice, and, if approved by the Advisory Council, update them,
   (b) Identify the need for new procedures and outline the purpose and content of these,

(c) With the support of the Specialist Committee on Uncertainty Analysis, review and if necessary amend, Procedure 7.5-02-05-03.3 "Waterjets - Uncertainty Analysis Example for Propulsion Test" to bring it into line with the ISO approach adopted by the ITTC,

(d) Include procedure for testing of bollard pull in Recommended Procedure 7.5-02-03-01.1.

3. Identify the parameters that cause the largest uncertainties in the results of model experiments, numerical modelling and full-scale measurements related to propulsion.
4. Check the possibility of adopting the findings of the Powering Performance Committee of 25th ITTC for improving the ITTC-78 method.
5. Follow developments in the field of podded propulsion with a view addressing the lack of model-scale and full-scale data in the public domain noted in procedure 7.5-02-03-01.3, "Podded Propulsor Tests and Extrapolation". Investigate the possibility of improving the procedure including separating it into logical parts such as resistance, propulsion, and extrapolation. Liaise with the Resistance Committee.
6. Comment on the impact of developments of propellers for ice going ships in the view of the increasing operations in ice covered waters and changes in regulations.

1.3 General Remarks

The task 2(c) was moved to Special Committee on High Speed Ships as recommended by Advisory Committee. The Committee asked Advisory Committee to clarify the scope of the ITTC Recommended *Procedures relevant to propulsion*. Also the Committee reported its opinion to focus on the Sections on the conventional propulsion and not to include the Sections on Cavitation, Ice, and High Speed vehicles listed below.

7.5-02-03-03: Propulsion/Cavitation
7.5-02-04-02.2: Ice Testing/Propulsion Tests in Ice
7.5-02-05-02: High Speed Marine Vehicles/Propulsion Test

Related with the tasks 2(a) and 2(b), the Committee distributed the questionnaire to find the need for new procedures from the member organizations. The analysis of the questionnaire is summarized in Section 3. The questionnaire is focused on the necessity of procedure for hybrid propulsors; however it seems premature to make any procedure.

2. UPDATE THE STATE-OF-THE-ART FOR PREDICTING FOR PROPULSION SYSTEMS EMPHASIZING DEVELOPMENTS SINCE THE 2008 ITTC CONFERENCE

Many major international conferences were held since the 25th ITTC conference in 2008;

- RINA Marine CFD 2008 (Mar. 2008, UK),
- 19th IAHR International Symposium on Ice (IAHR'08 July 2008, Vancouver Canada),
- 8th International Conference on HydroDynamics, (ICHD'08 2008, Nantes, France),
- 27th Symposium on Naval Hydrodynamics (Oct. 2008, Korea),
- 8th International Symposium on Particle Image Velocimetry (Aug. 2009, Australia),
- First International Symposium on Marine Propulsors - SMP'09 (June 2009, Norway),
- 7th International Symposium on Cavitation (CAV2009 Aug. 2009, U.S.A.),
- 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry (AMT'09 Sept. 2009, France),

- SNAME Propellers/Shafting'09 (Sep. 2009, U.S.A.),
- 10th International Conference on Fast Sea Transportation (FAST2009 Oct. 2009, Greece),
- 12th Numerical Towing Tank Symposium (Oct. 2009, Italy),
- 6th International Workshop on Ship Hydrodynamics (IWSH'2010 Jan. 2010, Harbin, China),
- 2010 International Propulsion Symposium (Apr. 2010, Japan),
- PRADS 2010 (Sept. 2010, Brazil),
- 28th Symposium on Naval Hydrodynamics (Oct. 2010, U.S.A.),
- 9th International Conference on Hydrodynamics (ICHD'10 Oct. 2010 Shanghai, China),
- 2nd International Conference on Advanced Model Measurement Technology for the EU Maritime Industry (AMT'11 April 2011).

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

2.1 Potential Impact of New Technologies on the ITTC

Currently, international maritime shipping accounts for 3% of the global CO_2 emissions and this value has been increasing. The International Maritime Organization (IMO) is working towards defining an Energy Efficiency Design Index (EEDI). This index will initially be a voluntary measurement used to compare the energy efficiency of new ships against an average performance of those launched between 1995 and 2005. The intent of the index is to stimulate innovation and technical development of all the elements influencing the energy efficiency of a ship from its design phase. While the details are still being developed by the international community, it is clear that restrictions on the production of CO_2 from ships will be developed and shall cause ship designers and owners to pursue new technologies to improve the energy efficiency of ships through studies on hydrodynamics and propulsion systems. Discussions on these topics can be found in Otsubo (2010) and the Climate Change and Ships Conference (2010).

Many papers have been published refining or combining previously known technologies using improved computational and experimental techniques to improve energy efficiency. This has enabled some technologies that previously have only produced marginal or inconsistent improvements to improve efficiency more reliably. An important element of assessing marginal technologies has been the ability to quantify the uncertainties in the model tests and the validated scaling procedures.

2.1.1 Azimuthing Thrusters The hybrid contra-rotating pod propulsion system has been successfully deployed on the Hamanasu, a high speed ferry, as described by Ueda and Namaguchi (2005). This concept is incorporated in a forward propeller driven by a conventional stern arrangement, while an aft propeller with a tractor pod acts as the second blade row of a contra-rotating propeller set.

The design and testing of this propulsion system has been the topic of recent papers by Sasaki (2009), Black and Cusanelli (2009). Sasaki discussed the design and complexities of open water and powering test of this arrangement. For open water testing, the forward propeller was driven by a propeller boat located ahead of the propellers while the pod dynamometer was installed aft as shown in Figure 1. A series of conventional open water tests and contra-rotating tests with propellers and dummy hubs were performed to establish the propulsor unit open water performance.

Figure 1 Open water test arrangement of hybrid CRP (Sasaki 2009)

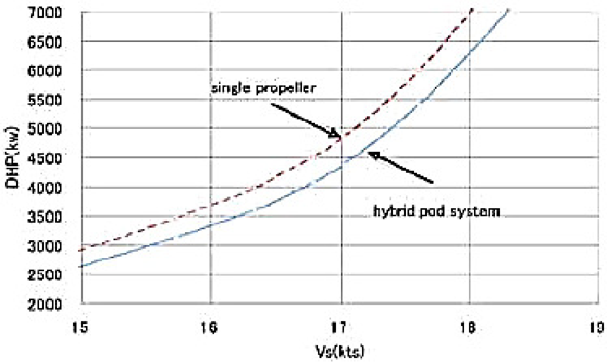


Figure 2 Comparison of Power(DHP) curves (Sasaki 2009)

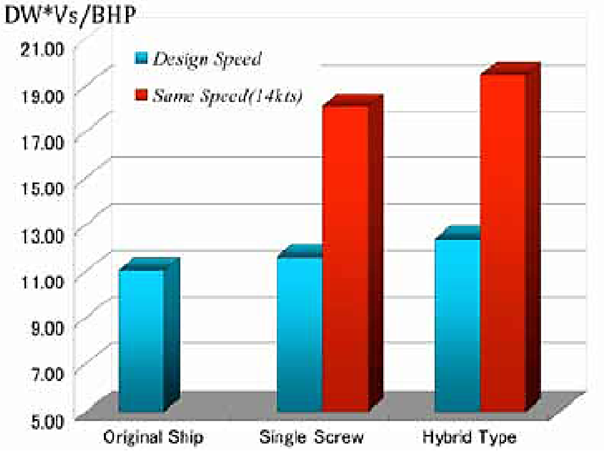


Figure 3 Economical evaluation of designs (Sasaki 2009)

In addition to the open-water test, resistance and self-propulsion tests were performed with conventional propeller-rudder configuration and hybrid CRP. The delivered power curves are shown in Figure 2. It was shown that the power difference is about 10% between single propeller and hybrid pod system at the design speed 17knots. Another important aspect of the design was the economical evaluation; 2 vessels and 2 design conditions were investigated as shown in Figure 3. A transportation efficiency defined as (DWT×Vs/BHP), taking transmission losses of electric drive into account, is compared. The testing at NSWCCD performed by Black and Cusanelli (2009) used a more traditional contra-rotating open water test with concentric shafts being driven by a downstream propeller boat as shown in Figure 4.

Figure 4 Open water test arrangement peformed for hybrid CRP (Black and Cusanelli 2009)

The NSWCCD approach does not include the gap between the fairwater and boss cap of the two propellers. Analytic calculations of the hub gap pressures are used in the NSWCCD approach to correct for the geometric inconsistence. Both groups suggest that using torque identity should be investigated as an alternative to the thrust identity procedure.

In addition to open water and powering testing, cavitation testing of a 35.5 cm / 28.1 cm set of contra-rotating propellers were performed in the NSWCCD 36” variable pressure water tunnel to determine cavitation inception and thrust breakdown through a thrust identity procedure using drive shafts from both upstream and downstream as shown in Figure 5 at a 42 knot condition.

Figure 5 Cavitation testing of a hybrid CRP in the NSWCCD 36" water tunnel (Black and Cusanelli 2009)

The NSWCCD experiment compares the efficiency of a pair of hybrid contra-rotating propulsors to a four-shaft propelled notional hull. The total delivered power to all four propellers of the Hybrid contra-rotating shaft-pod (HCRSP) configuration represented a reduction in delivered power of 14.7% at the threshold speed of 36 knots and 13.3% at the 39knot design speed, relative to the four-screw baseline. The maximum attainable ship speed, for using the envisioned total installed power, is 39.3 knots at 180 MW for the BSS and 38.97 knots at 150 MW for the HCRSP. To within experimental accuracy, the HCRSP is also capable of just attaining 39 knots. Incidentally, if the BSS were to be compared with the equivalent 150 MW total installed power, then its attainable speed would be reduced to 37.8 knots. A subsequent paper by Cusanelli (2009) compares these results to the same hull redesigned for axial and mixed flow waterjet propulsion.

The hybrid contra-rotating propulsion system was also evaluated by Takeda, Shimamoto *et al*. (2010), but with more of an emphasis on system engineering wherein issues with electrical generation, transmission and motor losses were accounted for to assess the hybrid CRP concept in terms of how much installed power would be needed for such a system instead of merely studying the hydrodynamics. For the container ship application being considered, the hybrid CRP was expected to have improved power requirements and improved maneuvering over a traditional single propeller.

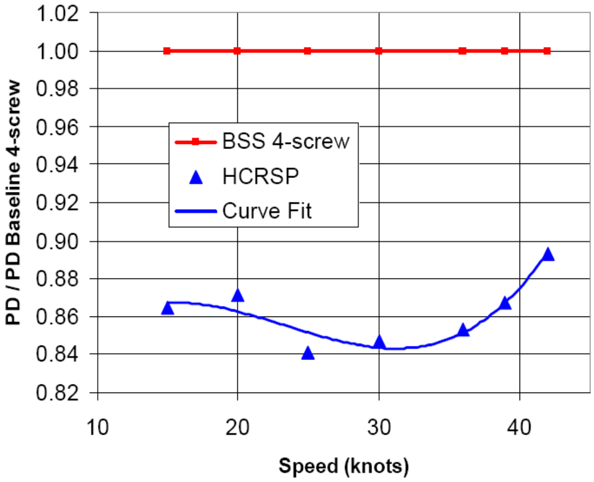


Figure 6 Delivered power comparison to BSS 4-screw baseline (Black and Cusanelli 2009)

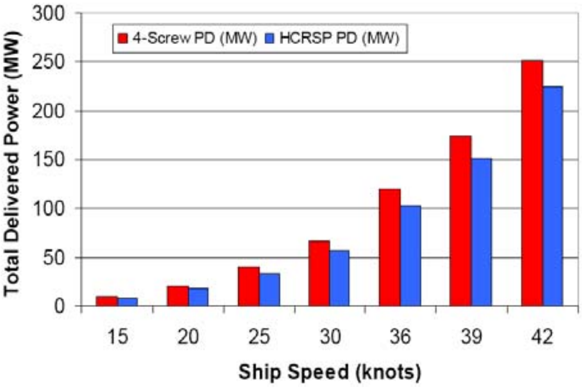


Figure 7 Delivered power comparisons (Black and Cusanelli 2009)

Funeno, *et al.* (2009) presented an application to the optimization of ducted azimuth thrusters using the commercial CFD software, STAR-CD. The effect of diameter and geometry of the gear housing on efficiency was investigated by CFD simulations. The predicted merit coefficient in bollard condition and open water efficiency in free sailing condition for the same ducted propulsor fitted with original and optimized gear housings were in good agreement with those measured.

Additional discussion of podded propellers can be found in Sections 2.2 and 2.2.4 where new experimental testing techniques are discussed. Section 2.4 where new benchmark data is introduced also discusses some pod model test data.

2.1.2 Alternative Material Propellers As the cost of metals worldwide increases, the development of propulsors made of alternate materials, such as composite glass and carbon fiber has become a topic of research around the globe. Depending on the materials and manufacturing process, the structural properties of alternate material propellers can be tailored to produce structures with rigid or pitch-adapting performance. The issue of fluid-structure interaction becomes important to designing, analyzing, testing and scaling the performance of these propellers.

The model testing and scaling of alternative material propellers is challenging. Pitch adapting composite propellers do not have a single open water curve to define their performance, since their geometry depends on dimensional loading. The thrust and torque characteristics of a composite model propeller will also not scale to full scale performance unless extreme care is taken in the selection of the materials and dynamic similarity characteristics. Work on this subject has been occurring worldwide by materials, structural, and hydrodynamic researchers, as reported in Lee and Lin (2004), Lin and Lin (2005), Blasques, *et al.* (2008), Young, *et al.* (2008), Tillmanns (2009), Yamatogi, *et al.* (2010), and Young, *et al.* (2010).

The need for capturing fluid-structure interaction for composite propeller has resulted in computational tool development specific to this problem as reported in Miller *et al.* (2010) using OpenFOAM RANS analysis coupled with a finite element analysis and by Young (2010) using a panel method coupled with a finite element analysis, as well as coupled commercially available software.

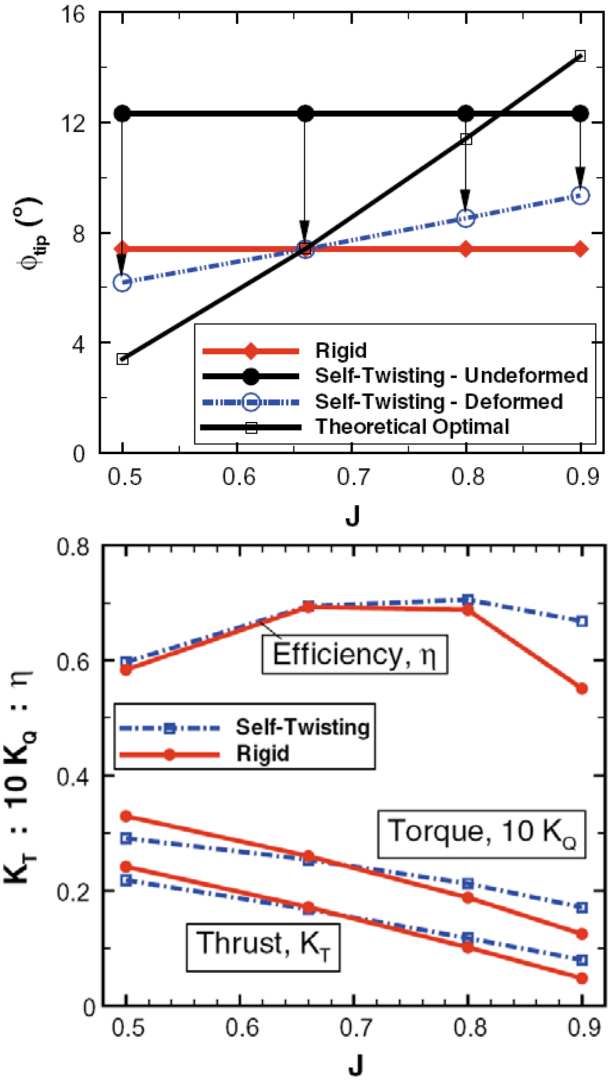


Figure 8 Comparison of the trip pitch angle (top figure) and performance curves (bottom) for the rigid and self-twisting propellers (Young, *et al*. 2010).

A reliability-based design and optimization methodology to improve the energy efficiency of self-adapting composite marine rotors was developed. It was shown that the uncertainties in material stiffness parameters, considered as random variables, have a marginal effect on the hydro elastic behavior of the self-twisting propeller. First-order reliability methods (FORM) were shown to be an adequate design tool instead of the more time consuming Monte Carlo simulations for probabilistic propeller optimization (Young, *et al.* 2010).

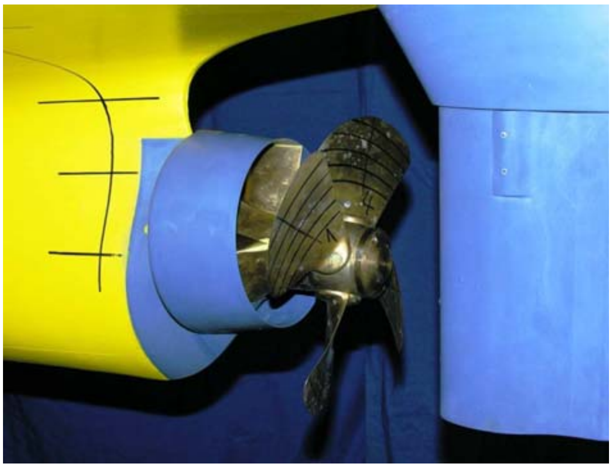


Figure 9 Mewis' pre-swirl duct (Mewis, 2009)

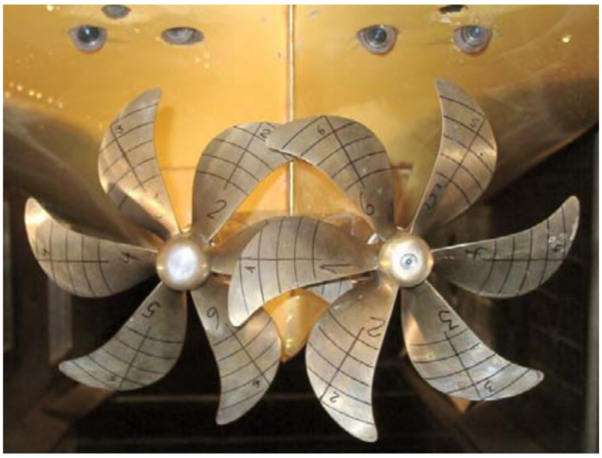


Figure 10 Kawasaki overlapping propeller system (Anda, *et al*. 2010)

2.1.3 Other Novel Propulsors The technologies discussed in this section are not necessarily new, but reflect current efforts to refine existing technologies with new computational and experimental techniques.

Chen, et al. (2010) made efforts to improve the efficiency of a 1,700 TEU containership through a series of hull form improvement and efficiency improving devices such as tip fin propeller, pre-swirl generator, boss cap fins, twisted rudder, rudder fin and rudder skeg. A combination of CFD and model tests at HSVA resulted in the study meeting its goal of a 10.3% power reduction through redesigning the transom, adjusting the ship trim and by using the best combination of energy saving devices. In this case, a tip fin propeller, twisted rudder, and rudder skeg were the best combination of energy saving devices. It is noted that scale effects on performance will need to be assessed through CFD and/or full scale trials.

Mewis (2009) presented a preswirl duct concept as an energy saving device for full form vessels with thrust coefficients greater than one and ship speeds less than 20 knots. The concept uses a wake influencing duct to accelerate additional flow to the inner radii of the propeller, while pre-swirl vanes inside of the duct add tangential velocity. The vanes are not axisymmetric and the duct is not vertically centered relative to the shaft. A combination of CFD and experimental results indicated around 7% reduction in required power.

Kawasaki shipping corporation and HSVA developed an Overlapping Propellers System that achieves high efficiency by generating pre-swirl into the propellers from upstream bilge keels (Anda, *et al*. 2010). The bilge keel vorticity is targeted to enter each of the overlapping propellers, shown in Figure 10. Rotational flow is also generated with horizontal bracket fins that extend between the propeller shaft and the centerline skeg. The paper describes the experimentally measured pressure forces from the overlapping propellers as being 30% of an equivalent single propeller design, as shown in Figure 11.

Hsin, *et al.* (2010) performed a computational and experimental study on propellers with tip shapes of the Kappel and CLT variety. The study used both panel code and RANS analysis for the propellers in uniform and non-uniform inflows to study their relative performance and assess scale effects. An example vorticity field of Kappel type propeller is shown in Figure 12. It was determined that CLT propellers are more sensitive to inflow variations than Kappel propellers. The scaling of the model performance to full scale needs to

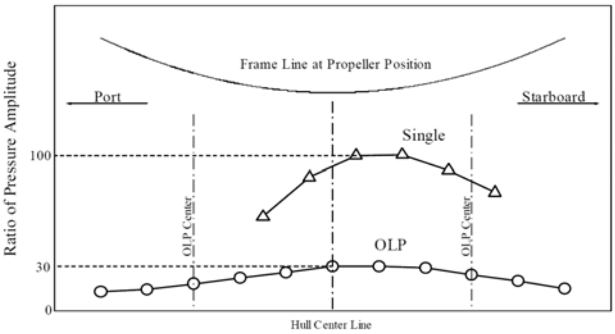


Figure 11 Comparison of Transverse Distribution of Fluctuating Pressure Amplitude (1st Blade Frequency) (Anda, *et al*. 2010)

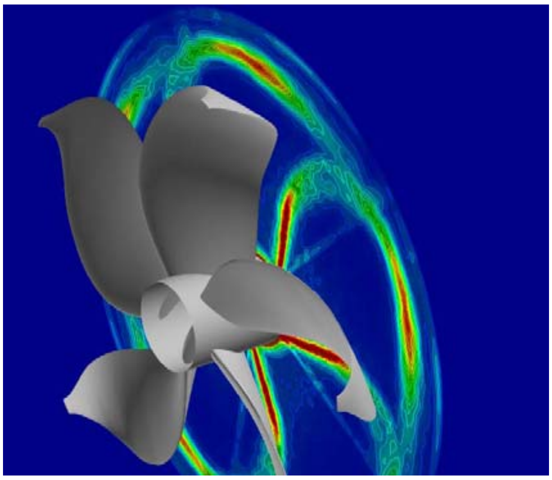


Figure 12 Vortex strength computed downstream of a Kappel style propeller (Hsin, *et al*. 2010)

be considered differently for these types of propellers than conventional propellers, particularly for torque.

Lee, *et al*. (2010) presented work on a wide chord tip (WCT) propeller that was designed, tested and tested at full scale. The wide tip allows for loading to be redistributed over the span of the blade without increasing pressure pulses on the hull. Efficiency gains of up to two percent are reported. The wide tips are highly skewed locally to the tip of the propeller, which has been shown to reduce cavitation generation and thereby pressure pulses.

Figure 13 IHI contrarotating propellers on a chemical tanker (Inukai, 2010)

Yamasaki, et al. (2009) suggested that increasing the hydrodynamic loading at the tip can increase efficiency without increasing the risk of cavitation erosion. This is achieved for propellers with short chords near the tips where any cavitation becomes super-cavities. The video clips of the cavitation captured by high speed camera and pressure pulse measurements were presented for a series of models together with CFD predictions to validate the evaluation tools.

Inukai (2010) presented a work done at IHI Marine United Inc. developing the diesel-electric propulsion vessels with Contra-rotating propeller (CRP). IHIMU adopted CRP and high efficiency hull form, shown in Figure 13. The effective wake factor (1-w) indicates the dramatic change compare to conventional propellers which sometimes reaches to the gain of 10% power reduction for a case with a forward CRP diameter equal to that of the conventional propeller installation. Both experimental and theoretical investigations were carried out to clarify the wake characteristics. Furthermore, the sea trials with a chemical tanker and an oil product carrier showed the 10% benefit in effective wake factor, which was almost equivalent to model test.

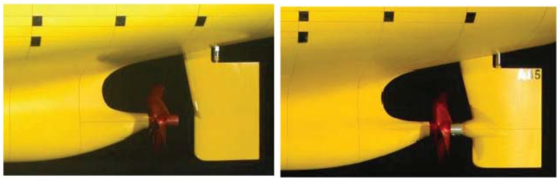


Figure 14 Rudder with and without end bulb (Oh, *et al*. 2010)

Oh, *et al*. (2010) identified the influence of the rudder bulb on thrust deduction, wake fraction and relative rotative efficiency through model testing of a containership with and without a rudder bulb, as shown in Figure 14..It was indicated that the ITTC wake scaling procedure may need modification for ships with rudder bulbs. It is suggested that the difference of wake fraction between with and without rudder bulb at model scale is added to that of full scale wake fraction prediction of no rudder bulb.

Jessup, *et al*. (2008) detailed powering analysis of two types of 4-waterjets high speed sealift models following the ITTC waterjet procedure. The relative differences between the axial and mixed flow waterjets are compared in terms of hull form design and powering performance. The experiment included detailed LDV measurements at Station 1, 3 and 6. The model arrangement is shown in Figure 15 for the axial flow configuration. The axial flow arrangement allows for a shallower and narrower transom, which resulted in improved powering performance at lower speeds, but at higher speeds, interactions between the inlets may explain its degraded performance relative to the mixed flow arrangement.

Figure 15 Four axial flow waterjet propulsion arrangement (Jessup, *et al.* 2008)

Karafiath, *et al*. (2009) introduced the Sea Train concept of high speed ocean transportation. This concept involves connecting multiple self-propelled small vessels together to create a single, long slender body with improved resistance and powering characteristics. Several different hull form types have been studied for this concept through experimental and analytical efforts with significant reduction in resistance. Inter-hull powering performance is a challenging aspect of the design and evaluation of this concept. An example configuration using three and four mono-hull notch-transom vessels is shown in Figure 16.

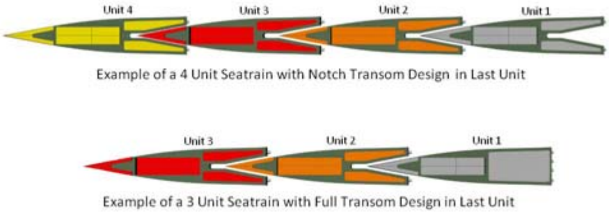


Figure 16 Example of a displacement hull Sea-Train concept (Karafiath. *et al*. 2009)

2.2 New Experimental Techniques and Extrapolation Methods

Hoshino, et al. (2010) observed propeller cavitation pattern by a high-speed camera system, synchronized with the propeller induced pressure fluctuation signals both for model-ship and full scale. Measurement was conducted for an 8,500 TEU container ship, which was built by the SHI. The ship is driven by a 6-bladed propeller with a diameter of 9.0 m. The model tests were conducted in large cavitation

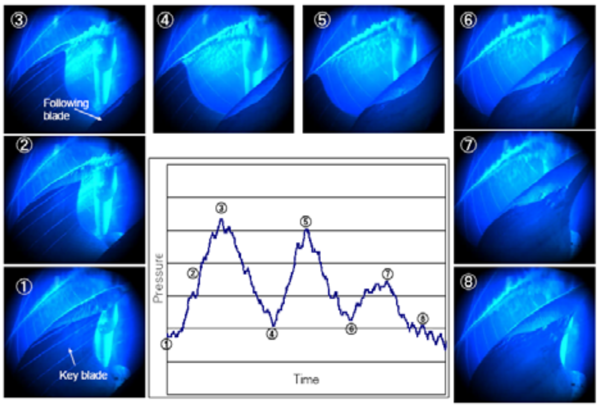


Figure 17 Relation between cavitation and pressure fluctuation signal in full scale (Hoshino, et al. 2010)

tunnel of SSMB (Samsung Ship Model Basin). The measuring section of this tunnel is rectangular with a length of 12 m, a breadth of 3 m, a height of 1.4 m and maximum velocity is 12 m/sec in the test section. A whole ship model was installed in the measuring section of the tunnel according to the full scale draught of 13.0 m even keel. The space between the ship model and tunnel wall was covered by wooden plates to suppress the free water surface. Cavitation observations and pressure fluctuation measurements were conducted at tunnel speed of 7m/sec and propeller rotational speed of 35.7 rps. For the full scale ship, pressure fluctuations on the hull surface induced by the propeller were measured with a pressure transducer installed on the bottom just above the propeller by using the so-called "bottom plug system". Acrylic observation windows of 300 mm diameter were also installed through the bottom of the stern to observe the propeller cavitation. The overall pressure fluctuation signals in both full and model scale are the superposition of the sharp pressure peaks due to the collapsing and rebounding of the tip vortex cavity on the key blade, and a gentle peak due to the growing and shrinking of the sheet cavity co-existing on the following blade. In general, the pressure fluctuation signal in the model test is similar to that in full scale. However, the minimum pressures in model test are lower and the maximum pressure peaks just after the minimum pressures are higher than those in full scale.

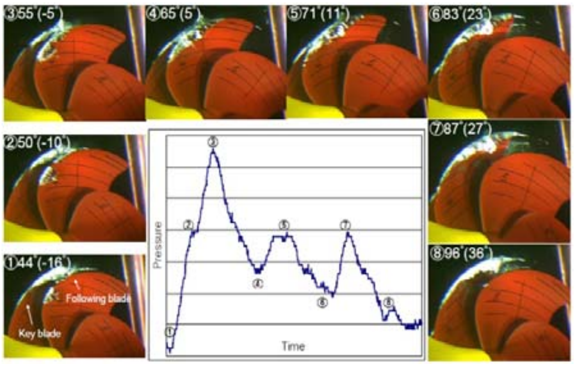


Figure 18 Relation between cavitation and pressure fluctuation signal in Model scale (Hoshino, et al. 2010)

Verhulst and Hooijmans (2010) addressed the problem of powering procedure for complex propulsion system of which example shown in Figure 19.

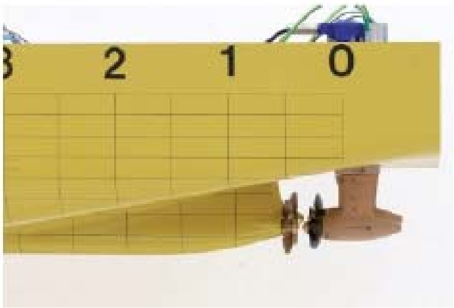


Figure 19 Twin hybrid contrarotating propulsor system (Verhulst and Hooijmans 2010)

They investigated a quasi-steady measurement technique, which has been also promoted by Schmiechen (2009). In this quasi-steady technique a gradual variation of the rotative speed of the propeller is imposed, while the forward speed of the model is kept constant. Thus, the load of the propellers continuously changes during the measurement run.

Although this paper indicates a need to develop a standard procedure for hybrid propulsion testing, there still remains a need of validation of the Quasi-Steady method through a comparison with full scale trials, which was not considered in this paper. Uncertainty analysis on this method would also be good to compare to the standard procedure which has not been performed. It is not suggested that a new ITTC procedure based on this approach be developed until it is more widely used and understood by the community.

2.2.1 PIV Measurements PIV (Particles Images Velocimetry) has become a widely used method for wake field investigation and propeller flows.

Fréchou, *et al.* (2009) reviewed various PIV techniques, applications and development that have been done in different European Marine Institutes for studies of wakes, propeller flows, wave breaking, and boundary layers. A benchmark measurement of PIV behind a flat plate has been carried out in different institutes to investigate the data discrepancies as shown in Figure 20. Borleteau *et al.* (2009) presented a benchmark measurement at a towing tank using a stereo PIVwith the surface piercing flat plate.

The main aim of the work is to test the efficiency testing of a Stereo PIV system. To achieve this work a flat plate (80 cm height and 50 cm chord) is vertically submerged to a depth of 30 cm; so the tip is near the free surface and it is possible to test the acquisition system in the presence of bubbles and other disturbances.

Many tests are performed at varying incidence angles (5, 20 and 40 degrees) and velocities of the flat plate in a towing tank. Grizzi, *et al.* (2009) used the Stereo PIV system with a fully submersible probe. The measurements give information on instabilities of the tip vortex of the plate. The intensities and geometries of tip vortices vary according to Reynolds number and the incidence angles. The test also comprises measurement planes along the chord at defined distances from the leading edge. The benchmark test might be carried out by international communities when advised by the ITTC Specialist Committee. The advantage of the benchmark test is that it allows the investigation of the overall uncertainties of PIV measurements in towing tank.

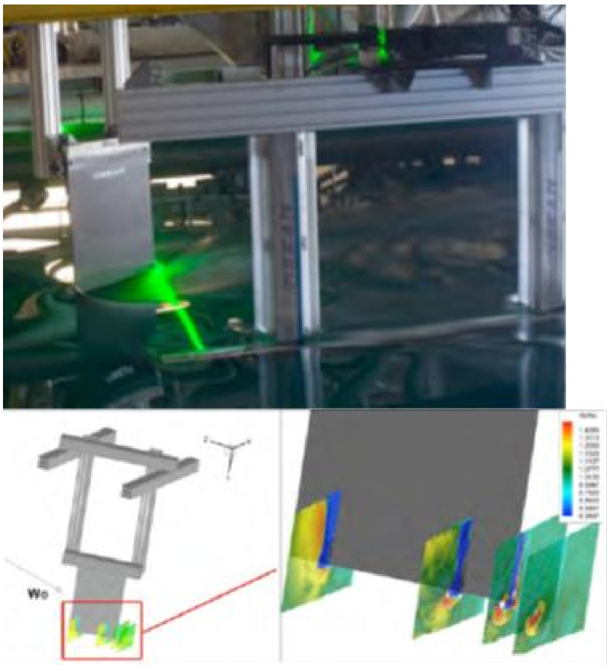


Figure 20 Flat plate benchmark setup in towing tank (Fréchou, et al., 2009)

Several papers are related to the use of PIV for propulsion testing in towing tank : Felli, *et al.* (2009); Nakaie, et al. (2009); Nagaya, *et al.* (2011); Kim, *et al.* (2011); Di Felice, *et al.* (2011).

Wu, *et al.* (2009) presented the use of PIV to investigate flow around the rotor of an axial hydraulic pump. PIV measurements are performed in the tip region of a water-jet pump rotor blade, in an optically index-matched facility, in order to study the structure and evolution of the flow. Data are used to examine the evolution of the tip leakage vortex (TLV) by means of the swirling strength to map the spatial distributions of multiple secondary vortices. Miorini, *et al.* (2010) presented 3D flow investigations, using high resolution PIV, on the internal structure of the tip leakage vortex within the rotor of the same axial waterjet pump. The paper provides detailed data on the instantaneous and phase averaged inner structure of the tip flow, and evolution of the tip leakage vortex, as shown in Figure 21.

2.2.2 Tomography PIV and Holography Techniques for 3D Flows Investigations

Thomas, *et al.* (2009) and Atkinson, *et al.* (2009) are investigating the performances of PIV techniques for the measurement of the three components of the velocity in a whole volume : Tomographic particle image velocimetry (Tomo-PIV) and Holography referring to a paper of Arroyo and Hinsch (2008). Although all these techniques are mainly applied to academic flows, they may be applied in the future to investigate the 3 components velocities in the flow volume around propulsors or in the inflow of the propulsor disk.

2.2.3 Manufacture of Model Propellers by Rapid Prototyping (RPT)

The manufacturing process of ship model components (i.e. rudders, propellers, fins, brackets, etc.), can be a time consuming and expensive procedure. For these reasons Bazzi and Benedetti (2009) have been investigating new manufacturing techniques known as Rapid Prototyping Techniques. These are aimed at reducing the cost and manufacturing time of the prototype hardware. A review of the possible applications of rapid prototyping techniques for the production of model ship components is presented. A comparison of the hydrodynamic behavior of similar propellers at model scale, made with different materials and produced using SLS (Selective Laser Sintering) technique, is reported. The open water tests were carried out at INSEAN towing tank for propellers of the E779 geometry to confirm whether the differently manufactured propellers are appropriate to use for the test.

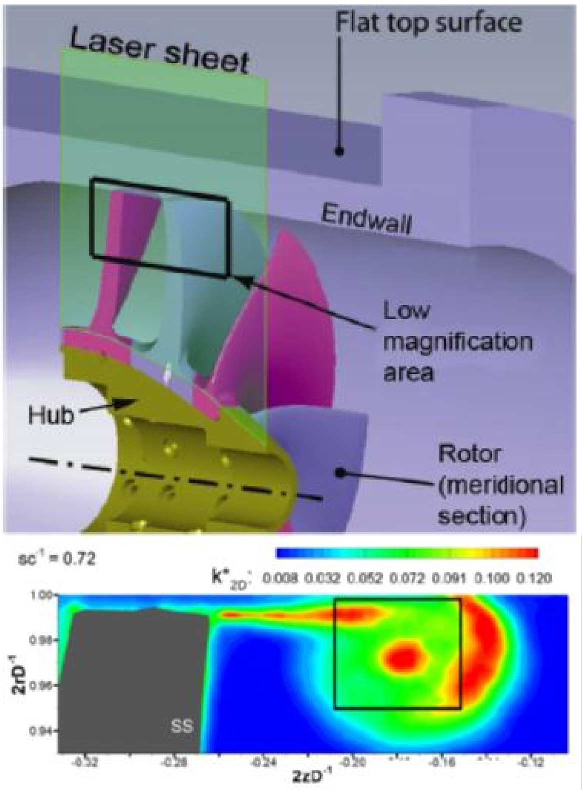


Figure 21 PIV measurements in a waterjet pump: turbulent knetic energy (Miorini, *et al.* 2010)

Figure 22 Bronze Propellers (Bazzi and Benedetti 2009)

Figure 23 Propeller in Sintered Metal powder (some machining imperfections are shown) (Bazzi and Benedetti 2009)

2.2.4 Lateral Forces and Non-Stationary Measurement on Propellers

A paper of Vartdal, *et al.* (2009) discussed full scale measurements of lateral forces from a ship propeller. In order to quantify the effects of the propeller forces on the shaft/bearing interaction, they have carried out a series of research projects where the lateral propeller forces have been measured directly on different vessel types in both steady state and transient operation. Analysis was carried out to investigate the effect of the propeller loads on the stern bearings of the vessel. The measurements have also been used as a benchmark to assess the predictive ability of current analytical methods. The results clearly illustrate the importance of being able to accurately predict lateral propeller forces in both steady state and transient conditions due to their significant effect on the bearing performance.

The amplitudes of the bending moment caused by blade order excitations are illustrated in Figure 24. The circles in the graph represent the variations of bending moments where the centre of the circle is the nominal bending moment, and the area represents the amplitude. The blade order amplitudes of bending moment variation are small to moderate.

Measurements were also carried out when the vessels were turning. The measurements showed a significant variation in the bending moments throughout the turns for all vessel types for starboard (Figure 25) and port (Figure 26) turns.

Islam, *et al.* (2009), Hagesteijn and van Rijsbergen (2009) reported non-stationary forces measurement on Pods. Both papers are presenting developments of test set-up for non-stationary force measurements. Although not much information is given on the dynamical capacity of the force transducers and Pod-Unit (main stiffness or first modal frequency), it is claimed that the tests set-up are able to measured dynamical forces frequencies up to the first blade rate.

Islam, *et al.* (2009) presented an experimental set-up to investigate the static and dynamic performances of an azimuthing podded propulsor (Figure 27). The paper reports on an experimental study into the effects of static and dynamic azimuthing conditions on the propulsive characteristics of a puller podded unit in open water. The model propulsor was instrumented to measure thrust and torque of the propeller, three orthogonal forces and moments on the unit, rotational speed of the propeller, azimuthing angle and azimuthing rate. The model was first tested over a range of advance coefficients at various static

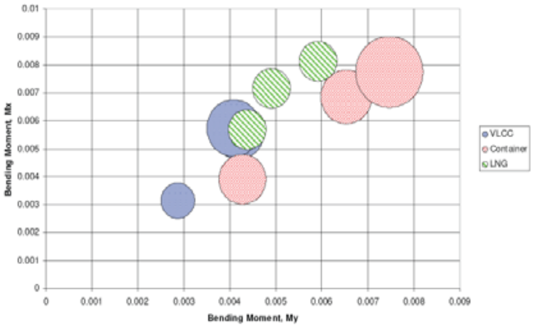


Figure 24 Bending moment amplitudes in ahead running Mx & My on VLCC, Container and LNG propeller (Vartdal, *et al.* 2009)

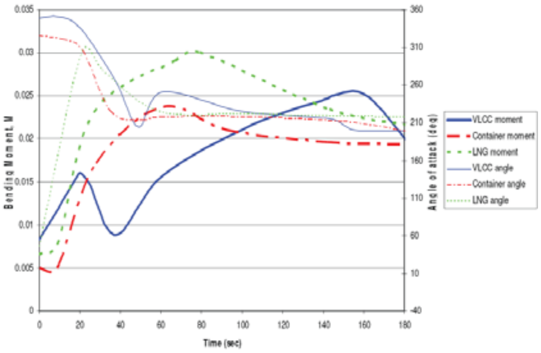


Figure 25 Bending moment during starboard turn (Moment as function of time) (Vartdal, *et al.* 2009)

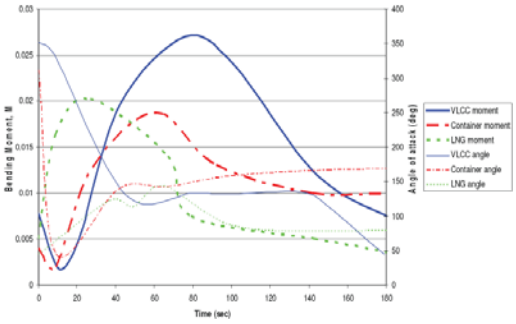


Figure 26 Bending moments during port turn (Moment as function of time) (Vartdal *et al.* 2009)

azimuthing angles, then the azimuthing angle was varied dynamically at different azimuthing rates and propeller rotational speeds. The performance coefficients of the propeller and the pod unit showed a strong dependence on the propeller loading and azimuthing angle (Figure 28). An uncertainty analysis of the measurements is also presented

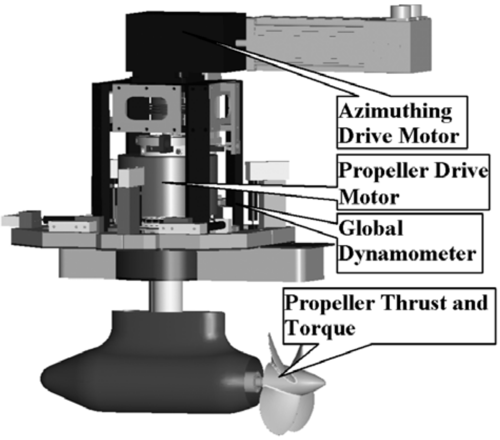


Figure 27 Test set-up for non stationary forces on measurements on pods unit (Islam, *et al*. 2009)

Hagesteijn and Rijstbergen (2009) reported on a new six-component propeller shaft balance for force measurements on pods. The main goal of the work was to develop CFD calculations to determine the loads on pods for various operational conditions, at the design stage. A specifically model test set-up has been designed to validate the CFD calculations. To determine the loads on the pod slewing and propeller shaft bearings, the model test set-up had to be capable of measuring six components (three forces and three moments) at the pod steering axis and at the propeller shaft. The force measurements device is capable of measuring the average forces and moments, as well as the unsteady forces and moments up to the first blade frequency. In addition, precise azimuthing angles and negligible mechanical vibrations were needed. The six component force dynamometer is in the propeller shaft between the motor seal and the propeller hub. It is claimed that the full six components of the propeller loads can be measured with the same accuracy as standard thrust and torque sensors.

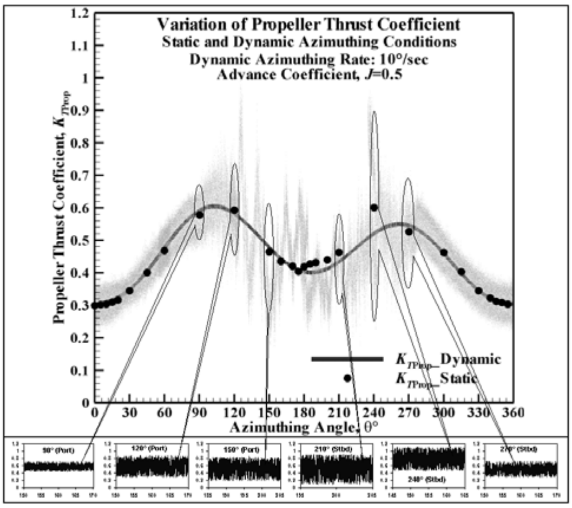


Figure 28 Experimental results: comparison of propeller thrust coefficient of the model pod unit at static (black solid circle) and dynamic azimuthing conditions (black dots for raw unfiltered data and black solid line for 10th order polynomial fit to the raw data).(Islam, *et al*. 2009)

Jessup, *et al*. (2009) developed an in-hub blade dynamometer to measure dynamic blade loading. To improve the prediction of the alternating blade loading under real operating conditions, a test program was conducted to measure the alternating blade forces in inclined flow in a water tunnel where effects of cavitation could be assessed. For these tests, an in-hub blade dynamometer was used along with a downstream slip ring housing, as shown in Figure 29. Instantaneous load variations were also measured to quantify peak transient loads due to cavity collapse (Figure 30 and Figure 31). Loading excitation due to strut wake turbulence was also identified. The LDV measured inflow was used to compare load predictions to the measured results.

Figure 29 Exploded view of propeller hub (Jessup, *et al*. 2009)

From the review of papers on non-stationary force measurement techniques, it can be conclude that, although there is a real need for such measurement techniques, more insight on the stiffness and the first modal frequency of the experimental set-up needs to be assessed. Not much is written in the reviewed papers on that point, which is very important to determine the real bandwidth of the force measurements technique.

2.2.5 Wireless Communication Wireless communication between sensors in ship models and amplifiers on land is becoming quite mature. Free running model tests offer complementary information from captive model tests. The model-ship can be free of restraints and unnecessary external forces and interference can be removed in free running tests.

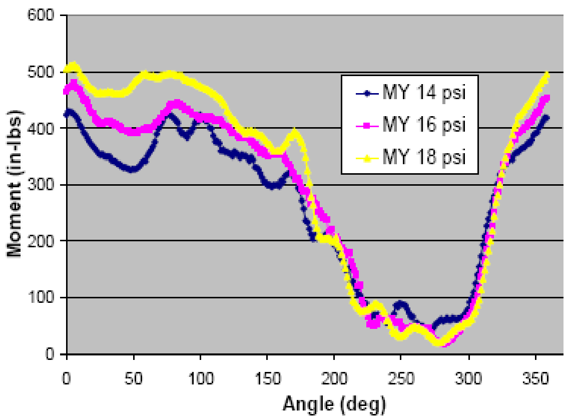


Figure 30 Variations in blade spindle torque, My, with blade angle at varying tunnel pressures, 1698 rpm, J=1.12 (Jessup, *et al*.(2009)

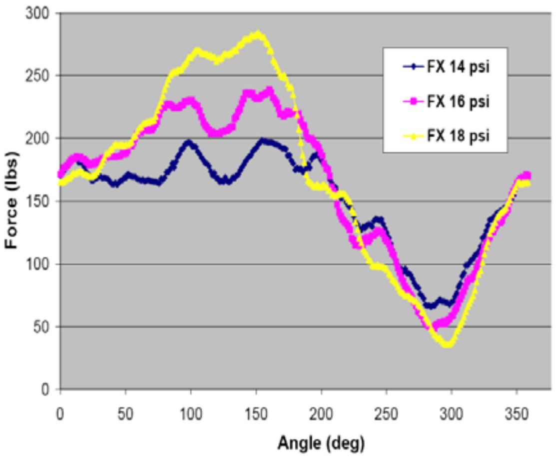


Figure 31 Blade thrust , Fx w/o DC correction (Jessup, *et al*. 2009)

Realtime wireless data and control communication of free running submerged scale models of submarines have been investigated by Kimber, *et al.* (2009). Wireless underwater communication is used to control the model in a maneuvering tank of QinetiQ. The experiments are truly free-running; no umbilical is used. The technology implemented is a radio system that works through the air-water boundary; it is not affected by electronics (including motor drives) in the submarine. This technology provides the capability to undertake maneuvering tests to investigate the performances of the propulsion and rudders combinations.

La Gala, et al. (2011) developed a setup to measure the torque acting on the blade as a function of pitch angle, in particular around the equilibrium points at different speeds (see Figure 32). The solution adopted is based on a wireless custom acquisition board placed inside the rotating shaft. The torque is measured using a miniaturized set of strain gages mounted on customized blades' joints. Data storage and transmission tasks have been achieved using an integrated on board memory and a 2.4 GHz transmitter module.

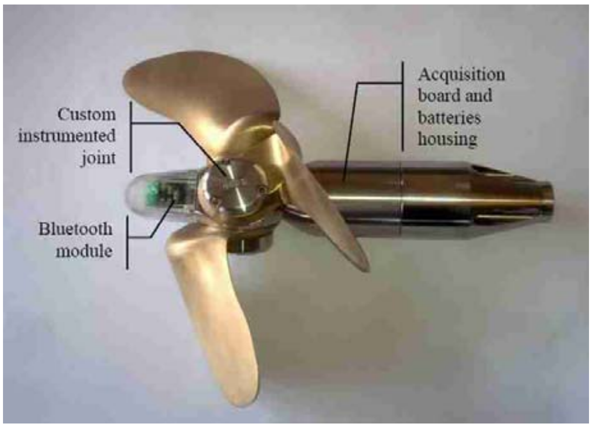


Figure 32 Wireless sensor for blade torque measurements (La Gala, *et al.*, 2009)

2.3 Extrapolation and Scaling Methods

As mentioned in section 2.1.3, Oh, *et al.* (2010) and Hsin, *et al.* (2010) both indicate that there may be a need to update the ITTC scaling procedures for rudder bulbs and for Kappel or CLT propellers. Both suggest alternative scaling methods for wake fraction and torque, respectively.

Chesnakas, *et al.* (2008) presented a test with a mixed flow waterjet at a range of scales and test facilities. Testing was performed with rotor tip Reynolds numbers varying from 3×10^5 to 10^7 in different facilities. Towing tank, pump loop and trials on a quarter-scale ship demonstrator are compared. Differences between the inflows for the different configurations make drawing conclusions regarding the most representative scale difficult.

Müller, *et al.* (2009) presented an analytical study of the scale effects on propellers for large container ships, which was performed by three institutions under the support of the German government. The advantage of this new method is the considering of the local 3-dimensional flow conditions. Using the ANSYS-CFX software, CFD simulations of the open water performance were conducted at model and full scales for 23 propellers covering a considerable range of blade geometry. The SST turbulence model was applied in combination with the transition model at model scale, and in its standard form at full scale. By analyzing the numerical results, extrapolation formulas were developed which account for the radial distributions of differences in model and full scale forces and their directions, as a function of the computed force/direction differences, skew, area ratio per blade, pitch ratio, and the thrust loading coefficient. To make use of the formulas, the radial distributions of forces and their directions as well as the open water test data are needed. An example was given in comparison with ITTC and Meyne's methods (see Figure 33).

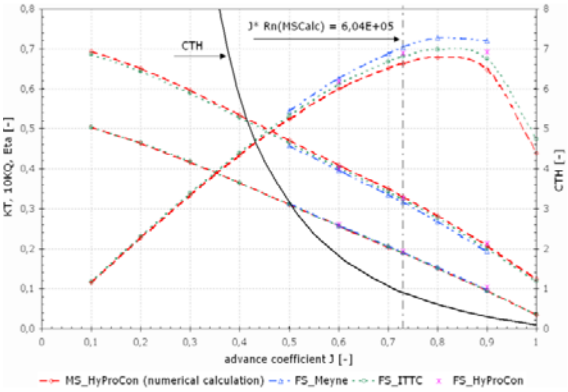


Figure 33 Comparison of predicted open water performance at full scale by different methods (Müller, *et al.*, 2009)

In the method, the scale effect mainly causes an increase in thrust rather than a decrease in torque as predicted by the other two methods. The open water efficiency predicted by the authors' method was in between those by the other two methods.

Krasilnikov, *et al.* (2009) investigated the influences of blade skew, loading, and area ratio on the scale effect of propeller by CFD simulation using the FLUENT software. The SST k-ω model was adopted for turbulence closure, though fully turbulent flow computations were carried out at both model and full scale. Three four-bladed propellers with 0°, 31°, and 62° balanced skew respectively and otherwise identical geometry (except for the thickness at outer radii of the most highly skewed propeller, Blaurock, *et al.*, 1988) were used as test cases. The Reynolds number at model and full scale were about

6.3×10^5 and 2.0×10^7 respectively. Unstructured tetrahedral cells were used near the blades, and prismatic layers were generated in the boundary layer region when possible. The mesh size was determined according to the verification study by Krasilnikov, *et al*. (2008). The predicted open water performances at model scale agreed well with experimental data, especially for the skewed propellers. Numerical results indicate that, from model to full scale, higher skew and lighter loading cause more increase in thrust and efficiency, and higher skew and heavier loading cause less decrease in torque. As compared with the CFD results, the increase in open water efficiency at full scale is more under-estimated by the ITTC'78 method when the skew is higher, mainly because the ITTC'78 method only accounts for the change of frictional drag with scale and does not include any correction for the skew. For a further study, the blade area ratio of the aforementioned propellers was enlarged to 0.7 from the original 0.5, while all the other geometric parameters were kept unchanged. It was found that, generally, the scale effect on open water performance became larger when the blade area ratio was increased.

Kawamura and Omori (2009) presented a CFD study of the scale effect on propeller open water performance for the 5-bladed Seiun-maru-I-CP and the 4-bladed propeller, MP282. The k-ω SST turbulence model and its low Reynolds number version (referred respectively as the high and low Re models below) were adopted in the simulations using FLUENT. For both propellers the open water efficiency predicted by the low Re model agrees better with experimental data at $Re\approx3.5\times10^5$ than that by the high Re model. Simulation results for the two propellers at several Reynolds numbers in the range of about 3×10^5 to 5×10^7 indicate that, the high Re model predicts a consistent increase of thrust and decrease of torque with increasing Re, which agrees reasonably well with experimental data. The results from low Re model, however, shows unclear trends for both propellers at $Re<10^6$. Further analysis shows that scale effects exist in both pressure and frictional forces. Both forces contribute to the scale effect in thrust, while the frictional force plays a major role in the scale effect in torque.

Bose and Molloy (2009) discuss factors that influence the extrapolation of model tests and make suggestions regarding how the present methods can be improved. They discuss in their presentation all items regarding the powering prediction process and make a proposal for a new method. The following main proposals are pointed out.

- More accuracy is expected obtaining the thrust deduction fraction from load variation tests. Those values have to be marked because they cannot be directly compared with traditional values.
- It is proposed not to use wake fraction and relative rotative coefficient for the extrapolation. More accuracy is expected by scaling the propeller coefficients in the behind condition compared with an allowance for the scaling of the wake. Research is necessary for such a procedure.
- Using the most accurate friction line (Grigson and Katsui are mentioned)
- If a form factor is used, then it should be taken from a regression formula with ship form coefficients as parameter, found by experimental data.
- Consequently a new correlation allowance has to be developed.

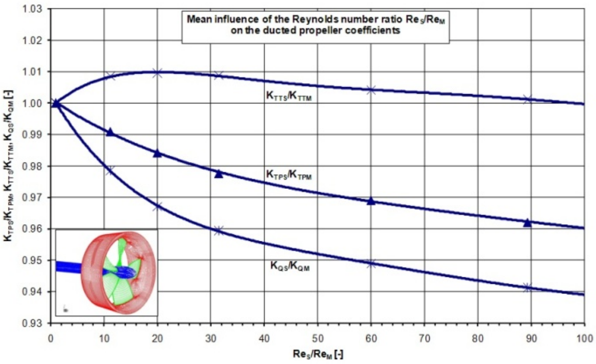


Figure 34 Influence of the Reynolds number on ducted propeller coefficients (Mertes and Heinke 2008)

The paper by Mertes and Heinke (2008) addresses ducted propellers and ducted rudder propellers performance scaling. One topic is the scale effect on ducted propellers. Based on CFD calculations and full scale test results a diagram (Figure 34) was developed which shows the change of ducted propeller coefficients in dependence on the Reynolds number ratio between full scale and model scale. These calculations were done for a propeller of type KA 5-75 in a nozzle 19A.

Furthermore, an estimation of the risk of bollard pull loss due to cavitation is given in this paper and reproduced in Figure 35. Based on CFD calculations a diagram was created which shows if a risk of bollard pull reduction may occur or not. In a case that such a risk exist, bollard pull tests under cavitation similarity are recommended in a depressurised tank or cavitation tunnel.

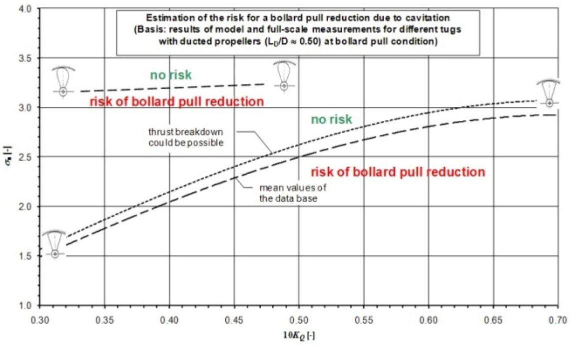


Figure 35 Risk of bollard pull reduction due to cavitation (Mertes and Heinke 2008)

2.4 New Benchmark Data

Several workshops and comparative testing or computational predictions with regards to propeller or propulsion have been recently held.

Glodowski, *et al*. (2009) presented a study focused on standardization of the testing procedures regarding pods performance. The joint procedure and benchmark tests have been proposed in relation with a preceding comparative study (ABB case presented by Veikonheimo, 2006), which exposed significant discrepancies between the results of pods related investigations. The experimental setups used at different facilities to test the same pod are shown in Figure 36.

Figure 36 Illustration of testing procedure (Glodowski, *et al*. 2009)

The paper is discussing the causes of the discrepancies found in the case of ABB and first of all on the results of the open water tests (differences of 5.9% in efficiency).

Savio, *et al*. (2011) presented the comparison between the experiments carried out by different model basins and numerical computations carried out by two different institutions have been presented. Though results are in most cases satisfactory, discrepancies still exists. The discrepancy between the different CFD calculation results is larger than the spreading in experimental results between the different model basins. From this, one might conclude that experiments still give more reliable results than CFD. This is even more so, since the CFD calculations were performed after getting access to experimental data. The discrepancy between experimental results and CFD is larger for full scale. However, in this case one can hardly conclude that the problem lies with the CFD – it could just as well be inaccuracies in the scaling method. Defining best practice for CFD analysis could be an interesting topic for future activities.

Streckwall and Salvatore (2008) are reporting the results of a workshop on propeller open water calculations including cavitation, held within the framework of European Research Project called VIRTUE. The workshop was addressing the computation of propeller flow under non cavitating and cavitating conditions for homogeneous inflow. nine different viscous codes (eight RANS – Comet (HSVA); FreSCo (MARIN, HSVA, TUHH), ISIS (ECN), STARCD (Wartsila), Fine/Turbo (NUMECA), FINFLO (VTT), Fluent (BEC),M-Uncle (ARL)- and one LES - OpenFoam (Chalmers)) were used with various solutions for propeller grids. The main differences between all the codes concern the pressure-velocity coupling (pressure correction methods versus pseudo compressibility), the near wall treatment (analysis with wall functions versus resolution down to the wall), the turbulence model, and the cavitation modeling.

In the case of non cavitating flow computation, no major difference was found between codes. The differences are found inside the flow field on very limited parts of the blade (tip region). The detachment prediction of the tip vortex is considered sensitive to the near wall treatment. 'Comet' using wall function and 'OpenFoam' with wall modeled LES predict an early detachment of the tip vortex.

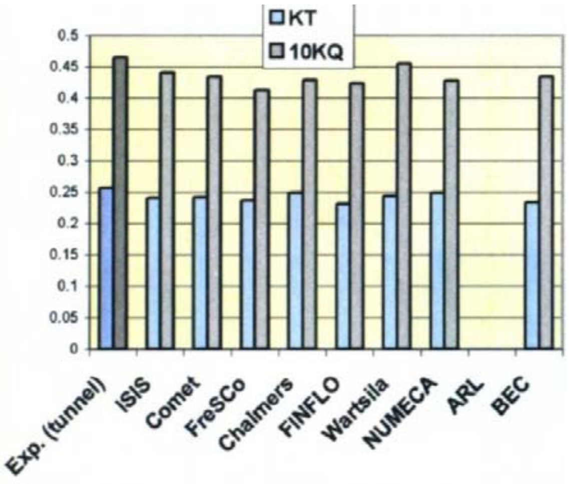


Figure 37 Prediction of thrust (KT), and torque (10KQ) at J=0.71 compared with measurements (Streckwall and Salvatore 2007)

One can note that neither the K_T nor the K_Q was ever over-predicted compared to experimental results, as shown in Figure 37. The differences between all the computations are of the order of magnitude of 5%.

The results from the cavitating case suggest that barotropic state law can be applied for steady state propeller cavitation as well as a transport equation based model. The cavitation models cannot be judged in view of the tip vortex cavitation capturing, since not too much of the tip vortex cavitation was resolved in any of the computations, as shown in Figure 38. They conclude that cavitation on propeller blades needs modified grids with sufficient resolution to resolve side jets that tends to lift the rear end of the cavity. They also point out that the task of post-processing propeller flow computations should not be underestimated and that within the workshop, practical aspects as working time for grid generation and CPU time for the computation were not discussed. They only state that they range from hours to several weeks.

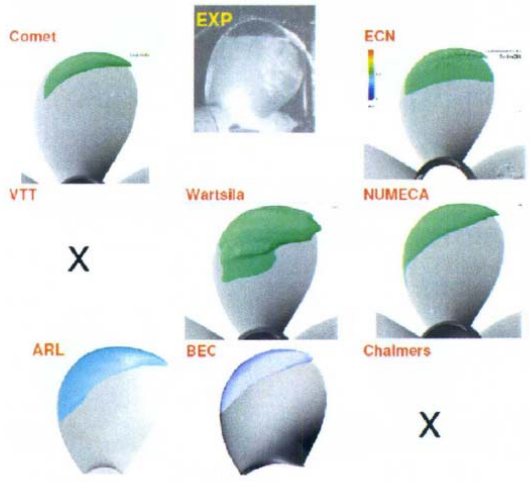


Figure 38 Iso surface for vapour volume fraction cv=0.5 at J=0.71, sn=1.76 (no results from FreSCo, FINFLO and Chalmers) (Streckwall and Salvatore 2007)

The model geometry and experimental data can be obtained from INSEAN after a non-disclosure statement is agreed.

At the Gothenburg 2010 Workshop on Numerical Ship Hydrodynamics, Larsson and Zou (2010) presented a summary and analysis of the submitted results of self-propulsion computations for the measurement of Kim *et al.* at Fn=0.26 only, as requested by organizers of the Workshop. In Case 2.3a the hull was kept fixed in its zero speed attitude, while in 2.3b the hull was free to sink and trim. Experimental data are available from NMRI for a 7.3m hull without a rudder in 2.3a, and from FORCE for a 4.4m hull with a rudder in 2.3b. Being consistent with the experiments, the computations were carried out at the *ship point* in 2.3a, and at the *model point* in 2.3b. There were 17 submissions altogether, of which 14 were for 2.3a and 3 were for 2.3b.

In the analysis the comparison error, E, its mean value, E_{mean}, and mean absolute value, $|E|_{mean}$, were used. $E=D-S$ in $\%D$, where D and S denote the measured and computed data values, respectively. It was found from the analysis (Larsson and Zou, 2010) that,

- There is a clear difference in scatter of E between the predictions having 10-24M cells and those below 10M. For K_T, K_Q and n the maximum scatter in the upper range is about ±7%, 5% and 2%, , respectively, while in the lower range it is within ±19%, 18% and 6%. For the towing force $R_{T(SP)}-T$, there are only 5 submissions and the largest error is for the largest grid (11.5M cells). All quantities but n have considerably larger errors than resistance.
- There is a clear trend of smaller scatter in E for the actual propeller in the K_T, K_Q and n plots. All three quantities have a smaller $|E|_{mean}$ for the actual propeller than for the modeled one, and the difference is particularly large for K_Q.
- The $|E|_{mean}$ for given n is only half of that for given *SFC*, while E_{mean} of K_T is somewhat larger for given n. If n is given the towing force is significantly over-predicted, while if *SFC* is given n is predicted very well.
- Since there are only 3 submissions for 2.3b, it is very difficult to draw conclusions by comparing the errors with those of 2.3a.
- By taking the weighted mean and standard deviation of 2.3a and 2.3b, a general indication of the accuracy obtainable in self-propulsion predictions may be given, as shown in Table 1.

Table 1 Mean error and standard deviation of self-propulsion predictions (Larsson and Zou, 2010)

	K_T	K_Q	n	$R_{T(SP)}$-T
Mean error	0.6%D	-2.6%D	0.4%D	-7.8%D
Standard deviation	7%D	6%D	3.1%D	8.7%D

2.5 Practical Applications of Computational Methods to the Propulsion Systems Predictions and Scaling

2.5.1 Hull-Propulsor-Rudder Interactions

Muscari, *et al.* (2010) presented a study of the flow around a propeller behind a fully appended hull by both CFD simulations and model experiments. The RANS solver developed at INSEAN was used. Measurements of the axial and vertical flow velocity components were carried out in the Large Cavitation Channel of INSEAN by means of a two-component back-scatter LDV system.

A comparison of numerical and experimental results was shown for a navy patrol vessel propelled by four-bladed twin CPPs in straight course, at $Rn=1.18\times10^7$, Fn=0.348 for the hull, and D=0.21m, Ω=820rpm, J=0.878 for the propellers. The flow around port side half of the model was calculated and measured. The real propeller geometry was adopted in CFD simulations. The simulations were conducted with a 12.7M fine grid and a 1.6M coarse grid respectively, and 50% of the cells were dedicated to modeling the propeller. For each time step the propeller rotates by a half degree.

It was demonstrated that the main features were correctly captured by CFD of the tip vortex interacting with the rudder. The CFD and EFD results were found to agree well for the axial velocity and transversal vorticity (see Figure 39) along the vertical mid-plane of the rudder, and for the axial velocity in two transversal cuts behind the propeller and rudder respectively. Along the trajectory of the tip vortex in the vertical mid-plane of the rudder, the axial velocity simulated with both fine and coarse grids was quite close to those measured, while the transversal vorticity was over-predicted especially with the fine grid.

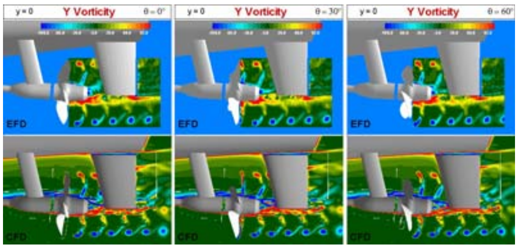


Figure 39 Comparison between EFD and CFD: transversal component of the vorticity field (Muscari, *et al*., 2010)

Han, *et al.* (2008) presented a CFD study of hull-propeller-rudder interactions using the software SHIPFLOW. The flow around hull and rudder was computed by steady RANS method with an algebraic stress turbulence model, while the propeller was replaced by a body force model based on the lifting line method. A chemical tanker was selected as the test case, with the rudder arranged at two locations behind the propeller. A series of computations were carried out which include propeller-rudder interaction in open water, grid dependence study for the resistance and nominal wake of the bare hull at full scale, resistance of the bare hull model at a number of speeds, and finally, self-propulsion computations for hull-propeller and hull-propeller-rudder combinations.

For the propeller and rudder in open water, it was shown that the numerical method was able to simulate at reasonably good accuracy the effect of axial spacing between propeller and rudder. Out of five sets of grid with different densities, the one with 2.3M points which produced a reasonably well converged wake for affordable computer effort was chosen for further computations at model. In the self-propulsion predictions the thrust deduction was slightly under-predicted compared with experimental results, while the wake fraction agreed well with the measured ones. The delivered efficiency was about 5% underestimated mainly because of an overestimation of the propeller torque. The influences of rudder location on local flow were investigated numerically by comparison of limiting streamlines on the hull and rudder, axial and cross-flow velocities at the propeller plane (see Figure 40), and axial velocities in four transversal cuts along the rudder and behind.

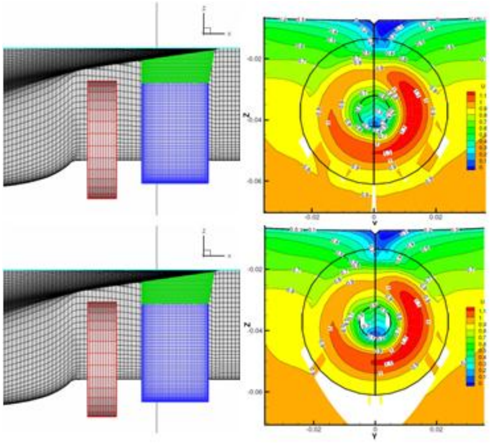


Figure 40 Comparison of simulated axial velocity contours at the propeller plane for two rudder locations (Han, *et al*., 2008)

Alin, *et al.* (2010) presented a study of submarine propeller-hull interactions based on the Large Eddy Simulation (LES) model. The open source software, OpenFOAM, was used to simulate, at model scale and in open water, the steady flow around the fully appended DARPA AFF8 submarine hull and the INSEAN E1619 propeller respectively, and the unsteady interactions between the hull and the propeller. In the unsteady case, the Deformation and Regeneration (D&R) method (Liefvendahl and Troëng, 2007) was employed.

In this method the computational domain is decomposed into three regions namely, V_f, the fixed region which encloses the hull and appendages, V_{rb}, the region which encloses the propeller and rotates with it like a rigid body, and V_t, the transition region in between V_f and V_{rb}. The grids in V_t are deformed as V_{rb} rotates, and regenerated when necessary. Based on computations for the E1619 propeller in open water done by Liefvendahl (2010), the fine grids are necessary in order to capture the features of wake flow induced by the trailing vortices. Coarse grids are also acceptable when the open water performance is of interest. The flows around the AFF8 hull without propeller and with the E1619 propeller in operation were computed by the LES method. The simulated hull-surface pressures and the axial velocity profiles in front of, and at the propeller disk plane were found to agree well with experimental data in both cases. The simulated variation of thrust for one blade at different angular positions was shown to discuss how the hull wake induces unsteady blade loading.

Carlton, *et al.* (2009) presented part of a CFD-based study on how the resistance and cavitation performances are affected by the rudder and head-box geometry, inflow condition, and the hull angle above the rudder, etc. The commercial CFD code STAR-CCM+ was used to simulate the flow around the rudder placed either in open water or in the propeller slipstream calculated using PROCAL, a BEM code for propeller developed at MARIN. Starting from a simplified symmetric spade rudder typically designed for a large container ship, the planar shape and thickness of rudder were altered by ‘tapering’ and ‘stretching’ while keeping the rudder surface area roughly constant. It was found that the ‘Longback’ rudder obtained by stretching the rudder aft of the widest point by a factor of 1.4 resulted in a negative drag in both the open-water and behind-propeller conditions, as well as better pressure distribution against cavitation. Besides, it was found from the test cases that a full head-box helped to reduce the rudder resistance greatly, so was the hull angle which was at least 10 degrees. Based on the CFD study, the authors concluded that pressure component dominates the rudder drag, and the best way to minimize it is to lengthen the rudder’s tail and reduce its thickness. Also it was found vital that the propeller slipstream should be considered when minimizing the rudder drag.

Lavini, *et al.* (2009) presented a wake-propeller design approach which demonstrates the potential of CFD tools for the purpose of optimizing the wake and reducing propeller excitation forces. By using the ANSYS CFX software, the hull and appendages were optimized to improve the wake quality, and the propeller designed by potential flow methods was further optimized to better adapt to the wake and avoid cavitation. The approach was applied to the design of hull and propeller for three passenger twin screw vessels. Through model test results it was confirmed that the propeller-induced pressure level and integrated excitation forces were much reduced without sacrificing the efficiency. The predicted vertical excitation forces were one order lower than those according to the van de Kooy criteria.

2.5.2 Propulsor/Hull Optimization Van de Ploeg and Raven (2010) used a RANS code, PARNOSSOS, coupled with a parametric hull form optimization tool to develop hull forms that both minimize resistance and seek to improve the wake quality into the propeller. This project developed from the VIRTUE program but with alternative objective function. In a sample problem that examines the afterbody of a tanker, the significant reductions in resistance were achieved with good correlation between the RANS and experimental data.

Hollenbach and Reinholz (2010) presented a summary of recent HSVA model testing and computational efforts to improve powering performance through hull form optimization and utilization of various energy saving propulsor devices. The paper provides a good reference as to the types of energy saving devices currently being considered for merchant ships.

Peri, *et al*. (2010) presented work on the optimization of the hull form and propulsor of a waterjet propelled monohull and catamaran. A combination of different fidelity models are used to solve a multi-objective design optimization problem. The URANS code, CFDSHIP Iowa, was extended to include the simulation of waterjet propulsion and adopted for the high-fidelity analysis. The wave resistance code, WaRP, developed at INSEAN with dynamic heave and trim capabilities was used for the low-fidelity analysis. The optimization was based on Global Optimization (GO) derivative-free algorithms.

A Joint High-Speed Sealift (JHSS) design, which is 970 ft long and operates at a transit speed of at least 36 knots using four axial flow waterjets, was selected for the verification and validation (V&V) of CFDSHIP Iowa, and as the initial geometry for subsequent optimization. The V&V study for the total resistance, trim angle, volume flow rate, inlet wake fraction, gross jet thrust, and inlet efficiency, etc. proved that the CFD code is an efficient and accurate tool to predict the waterjet-propelled JHSS.

The JHSS baseline hull bow shape was optimized by single objective (for total resistance) and multi-objective (for total resistance and seakeeping) optimization. The seakeeping function was calculated using INSEAN code FreDOM. The 7% reduction in resistance and the 4% reduction in seakeeping function compared with the original hull were achieved according to the numerical results.

A sensitivity study was carried out for JHSS WJ intake duct shape at Fr=0.34. About 35 cases were investigated by varying the upper curvature and lip shape of the inlet duct, and the optimal shape showed 0.23% increase in speed and 1.3% reduction in total drag for both types of modification as seen on Figure 41.

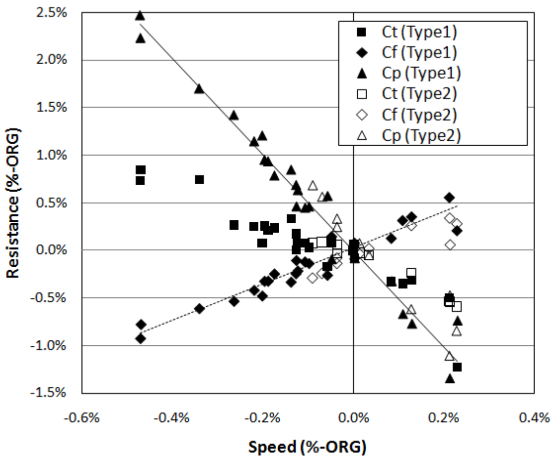


Figure 41 Results of sensitivity analysis for WJ intake modeling (Peri, *et al*., 2010)

The preliminary validation results were also presented for the bare hull resistance and self-propulsion of the waterjet propelled Delft Catamaran. The predicted resistance, thrust deduction, and sinkage agree reasonably well with experimental data, however there is a large discrepancy between predicted and measured trim at self-propulsion conditions.

2.5.3 Multi-Component Propulsors Sánchez-Caja, *et al.* (2009) presented a study of different CFD modeling approaches for the interaction between ducted propeller and rudder using the FINFLO code which solves the RANS equations by FVM and block-structured grids. The comparison of the open water performances obtained from steady (mixing plane), quasi-steady (MRF), and unsteady (sliding mesh) simulations at model scale shows that, for the configuration considered, the steady and quasi-steady results were both quite close to the unsteady results in terms of average thrusting forces at component level. The quasi-steady and steady approaches were better in predicting the forces and efficiency respectively.

Taketani, *et al.* (2009) presented a CFD application to the design of a ducted propeller with better bollard pull performance. The commercial CFD solver, STAR-CD, was used to investigate the influences of the section profile of duct and the pitch distribution and disk location of propeller blades on the thrust and merit coefficient in bollard condition. The numerical approach was proved effective by model and full scale tests.

Minchev, *et al.* (2009) presented a design case study for a 120t bollard pull off shore support vessel equipped with an Alpha High Thrust nozzle. In the design/optimization approach, CFD plays an important role in optimizing the aft ship hull form, nozzle and propeller geometries, as well as shaft strut and nozzle/rudder supports. A significant increase of the bollard pull was verified by model and full scale tests, showing that CFD is an effective and economical tool for such applications.

Sileo, *et al.* (2009) presented a CFD study to explain the reason for the large drop of turning moment for an anchor-handling towing service (AHTS) vessel equipped with twin ducted propellers and a tunnel thruster. RANS simulations of interactions among the main propulsors (MPs), the tunnel thruster (TT), and the hull were carried out using the FLUENT software for different loading combinations of the MPs and TT, where the propellers were all modeled by the actuator disk. The calculated thruster force and turning moment agree well with those measured in model tests (Sileo and Steen 2010) at Marintek. The simulated flow indicates that the thruster race is partially or totally sucked into one of the MP disks (Figure 42), which causes an additional turning moment in opposite direction to that of the TT due to the additional negative side forces and thrust asymmetry acting on the MPs. The CFD results also help to explain why the drop of turning moment becomes severe as the loading of MP is increased.

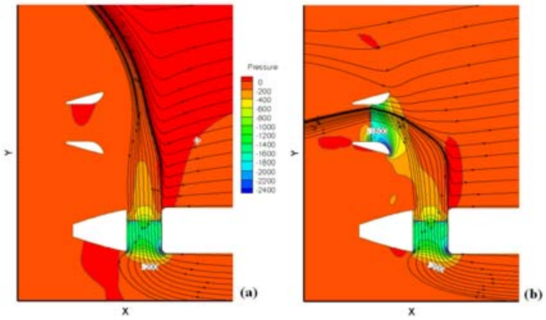


Figure 42 Action of the thruster race on the inflow entering the duct. Pressure contours and projection of streamlines on the horizontal plane passing in the middle of the tunnel. Fully loaded thruster. MP loading (a) 0 and (b) 35% MCR. (Sileo, *et al*., 2009)

Liu (2009) developed a potential-based surface panel method for predicting the steady and unsteady performances of contra-rotating propellers. The equal-pressure Kutta condition and an empirical trailing vortex model are implemented, and the mutually induced velocities are used to address the interaction between the forward and aft propellers. The method was validated against two sets of CRPs, DTMB 3686+3687A (4-0-4) and DTMB 3686+3849 (4-0-5), for which model test data in uniform flow are available (Miller, 1976). For both sets, the total thrust and torque predicted by the steady model agree quite well with the model test data, though the thrusts of the forward and aft propellers are somewhat over estimated and under estimated respectively. Such discrepancies are smaller in the results obtained from the unsteady model (at one advance ratio). For both sets the amplitudes of predicted unsteady thrust and torque at the lowest interaction frequency are about 1/2~2/3 of those measured for the forward propeller, and 2~4 times of those measured for the aft propeller.

Fujisawa, *et al.* (2010) presented a CFD modeling approach for the open water performance of contra-rotating propellers using the FLUENT software. The SST k-ω model with low-Re correction was employed for turbulence closure. The sliding mesh model was used to account for the interaction between forward and aft propellers. Two sets of CRPs with 4/4 and 4/5 blade combinations respectively were computed at Re=2.0~2.5×10^5. For both sets at equal and unequal rotational speeds, the predicted thrust and torque are generally in good agreement with experimental data, though the thrust of aft propeller is underestimated. The predicted open water efficiency is lower than the measured, especially for the 4/5 set which is moderately skewed.

Zhang, *et al.* (2008) presented their work on the prediction of podded propulsor performance by solving the RANS equations and the SST k-ω turbulence model using FLUENT. As an improvement to their previous work, prismatic cells and block-structured hexahedral cells were used around the propeller blades and the pod housing respectively to enhance the resolution to boundary layers. The quasi-steady (MRF, or 'frozen rotor') and unsteady (sliding mesh) modeling approaches were adopted to compute the open water characteristics of a podded propulsor model operating in both pulling and pushing modes and without the helm angle. Numerical results indicate that the predicted blade thrust agrees well with that measured, while the torque is over-predicted. The discrepancy was attributed to the lack of a transition model in the computation. The results also indicate that unsteady modeling is necessary to improve the prediction accuracy of the pod housing drag, particularly in pulling mode where the unsteady interaction between the propeller wake and the pod housing plays an important role.

Ma, *et al.* (2008) presented a method for predicting the open-water performance of podded propulsors based on potential flow theory. The propeller blades are calculated by a vortex lattice method, while the pod housing by a surface panel method for non-lifting bodies. The interaction between the blades and the pod housing is accounted for by taking the velocities induced by the blades as part of the inflow to the pod and vice versa. For puller-type podded propulsors, the propeller trailing vortex model is modified to account for the presence of the pod housing in blade wake. Iterative computations are performed until the hydrodynamic forces converge. Generic podded propulsors in pulling and pushing modes were taken as numerical examples. It was shown that the predicted blade thrust and torque agree well with those measured in cavitation tunnel tests over a range of operating conditions.

2.5.4 Crashback The performance of propellers in the crashback quadrant continues to be studied computationally using Large Eddy Simulations by Jang and Mahesh (2008, 2010) for open and ducted propellers. These calculations use unstructured grids to compute the thrust, torque and side forces and are compared to experimental data collected in a water tunnel for a range of advance coefficients in the crashback quadrant.

Berchiche (2008) demonstrated that the Large Eddy Simulation capabilities in the FLUENT commercial software are similarly capable of solving crashback flows. The calculations used a sliding mesh formulation to compute the flows around propeller 4381 from NSWCCD. A structured mesh formulation was used with one-million cells for the entire domain. The computed mean and standard deviations of crashback forces agreed well with measurements.

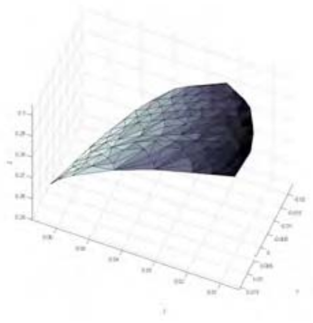


Figure 43 Propeller blade reconstruction (Savio, *et al.* 2009a)

The CFD work on crashback using large eddy simulations (LES) by Chang, *et al*. (2008) has also been coupled with finite element analysis to investigate the physics of high- and low-amplitude loading.

2.6 New Developments of Experimental and CFD Methods Applicable to the Prediction of Cavitation

2.6.1 New Experimental Cavitaion Prediction Methods Savio *et al.* (2009a) developed stereo imaging technique for cavitation structure measurements. The experimental setup and software tools currently under development in the cavitation tunnel of Genoa University are presented. Computer Stereovision is used to develop 3D reconstruction algorithms of the cavitation structure occurring on propeller blades (See Figure 43 and Figure 44). Currently a volume reconstruction method based on stereometry and an active stereo technique, with source light produced by a triggered laser, are studied. Both techniques are presented along with preliminary results, clearly outlining their merits and shortcomings.

In order to perform stereo matching (physical point well identified on both cameras images), images texture or some conspicuous points are required. Unfortunately, sheet cavitation does not provide, in general, any of them.

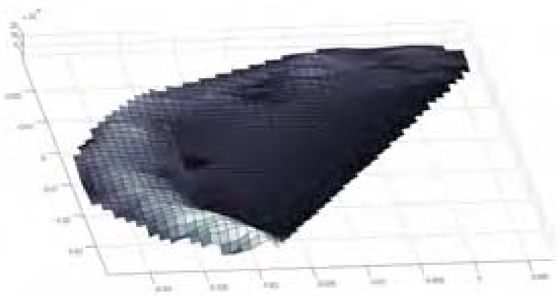


Figure 44 Sheet cavitation thicknesses (Savio, *et al*. 2009a)

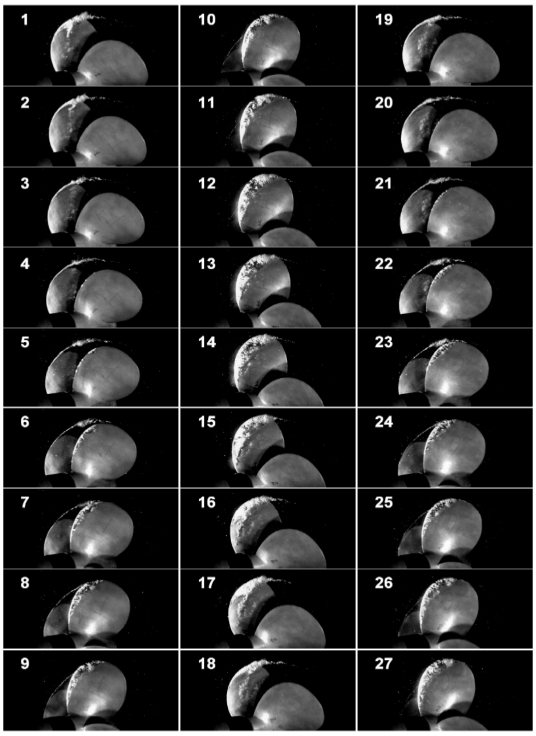


Figure 45 High-speed video images of sheet cavity on the propeller (Pereira, *et al*. 2009)

To overcome this problem a conspicuous point can be artificially created, and hence the technique is active, by projecting a light ray or a texture with a coded light pattern. This is the active stereo technique. Although the authors do recognized that further development are needed in order to become fully reliable, the technique seems very promising. This paper of Savio, *et al.* (2009b) is related to a paper of Pereira, *et al.* (2009) that provides guidelines for the use of High Speed Video (HSV) in combination with pressure and noise measurements.

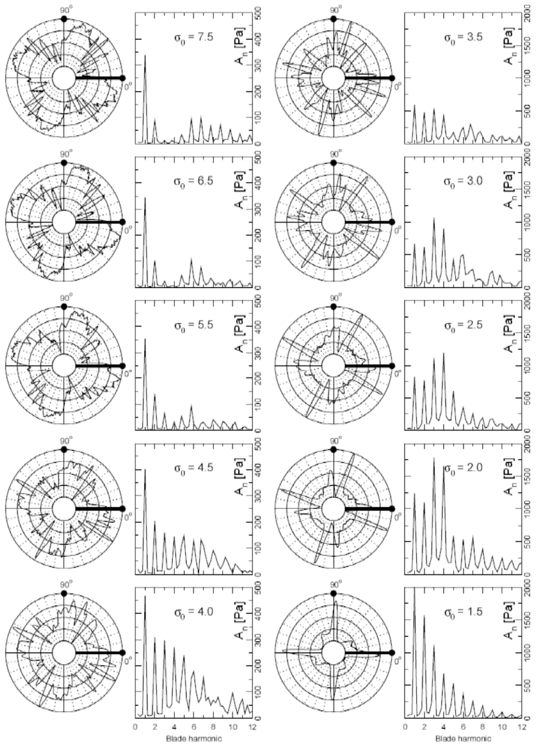


Figure 46 Simultaneous hull pressure pulses measurements with high speed video images of sheet cavity on propeller (Pereira, *et al*. 2009)

Pereira, *et al*. (2009), they reviewed the technical requirements for an adequate implementation of High Speed Video, as well as guidelines on best use and recommendations on the illumination and image preconditioning in order to obtain the optimal image quality for analysis. They also described practical implementations and cases of interest, which illustrate the procedures and methods used in different hydrodynamic testing facilities. Combined with the simultaneous data recording of characteristic physical properties such as pressure and noise, HSV can provide a new insight into the cavitation phenomena that are at the core of the vibration excitation of ship hulls and erosion of propellers and rudders (Figure 46).

A comparison of three nuclei measurements techniques has been presented by Mées, *et al.* (2010) and Mées, *et al.* (2011). The knowledge of the nuclei content is essential in naval hydrodynamics for cavitation inception prediction on propellers and hydrofoils as shown in the 21st ITTC Cavitation Committee report. Tip vortex cavitation which is, generally, the first cavitation to occur on foils or propellers, is very sensitive to the nuclei content. The papers present the comparison of three different types of techniques: the digital in-line holography technique and the Interferometric Laser Imaging Technique (ILIT), based on the PIV optical arrangement that measure the nuclei size, and Venturi techniques which is directly measuring the critical pressure of the nuclei (Figure 47).

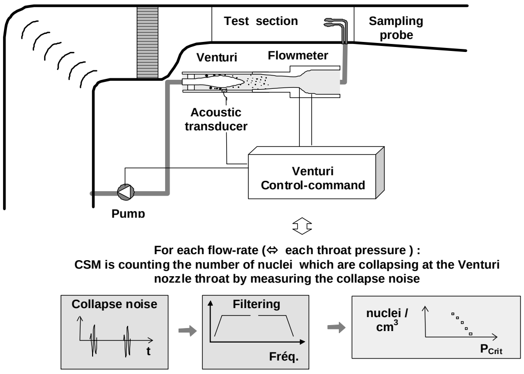


Figure 47 Venturi technique for critical pressure distribution of cavitation nuclei measurement

Although, the Venturi technique provides directly the real information on the critical pressure of the water, the Interferometric Laser Imaging Technique might be worth to be used since it is using the same equipment as for PIV except for the optic that could be adapted to very small bubble size (D<100µm) (Figure 48).

Figure 49 shows a comparison of the distribution of nuclei critical pressure using the ILIT method and the Venturi method. The measurements were performed in the French Large Cavitation Tunnel (Fréchou, *et al.*, 2000).

2.6.2 New CFD Prediction Techniques for Cavitation. Salvatore, *et al.* (2009) presented the results on cavitating propeller modeling from the VIRTUE 2008 Rome workshop. Seven computational models as listed in Table 2 were employed for a benchmark analysis of

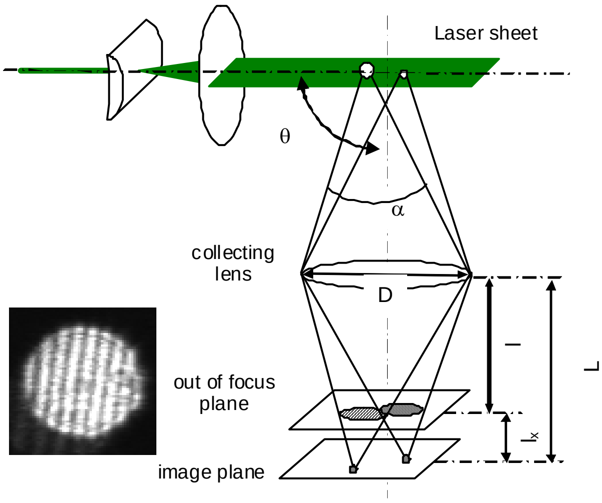


Figure 48 ILI Technique for size distribution of cavitation nuclei measurement

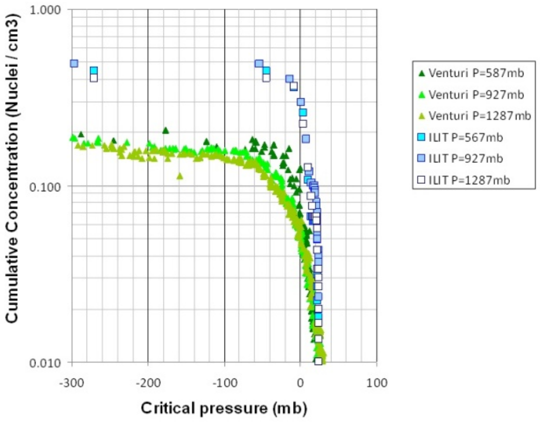
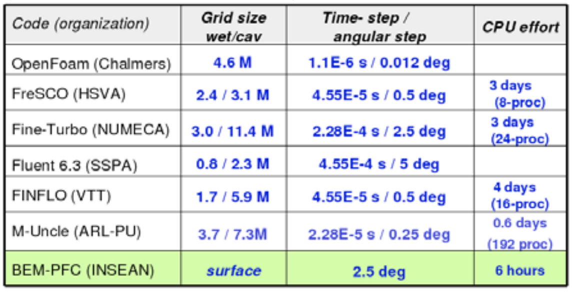


Figure 49 Comparison of nuclei critical pressure using ILIT and Venturi methods

the INSEAN E779A propeller model in uniform and non-uniform inflow conditions.

The main features of the computational models were described. A comparison of the open water characteristics predicted by the seven models with the open water test data was presented first. The errors in predicted thrust and torque fall within a range of -10% ~ +8%. Then the simulated cavity geometries in steady and unsteady conditions were compared among different models and with experimental recordings. Figure 50 shows a comparison of predicted unsteady cavity extents. As concluded by the authors, qualitative agreement was confirmed between numerical and experimental results, though quantitative differences existed in predicted cavity extent which could lead to differences in predicted pressure fluctuations and erosion.

Table 2 Summary of computational details (Salvatore, *et al.*, 2009)



Code (organization)	***Grid size wet/cav***	***Time- step / angular step***	***CPU effort***
OpenFoam (Chalmers)	**4.6 M**	**1.1E-6 s / 0.012 deg**	
FreSCO (HSVA)	**2.4 / 3.1 M**	**4.55E-5 s / 0.5 deg**	**3 days (8-proc)**
Fine-Turbo (NUMECA)	**3.0 / 11.4 M**	**2.28E-4 s / 2.5 deg**	**3 days (24-proc)**
Fluent 6.3 (SSPA)	**0.8 / 2.3 M**	**4.55E-4 s / 5 deg**	
FINFLO (VTT)	**1.7 / 5.9 M**	**4.55E-5 s / 0.5 deg**	**4 days (16-proc)**
M-Uncle (ARL-PU)	**3.7 / 7.3M**	**2.28E-5 s / 0.25 deg**	**0.6 days (192 proc)**
BEM-PFC (INSEAN)	***surface***	**2.5 deg**	**6 hours**

The benchmark analysis indicates that further study is needed of turbulence and cavitation models, grid resolution, numerical dissipation and, being a critical issue, the modeling of non-uniform inflow.

Sipilä, *et al.* (2009) presented a study of wetted and cavitating propeller flows in both uniform and non-uniform inflows using FINFLO, the RANS solver developed by Helsinki University of Technology. For incompressible flows, the FVM solver employs a 3rd-order upwind biased scheme to discretize the convection terms and incorporates Merkle's cavitation model. The low Reynolds number k-ε model of Chien was used in the computations.

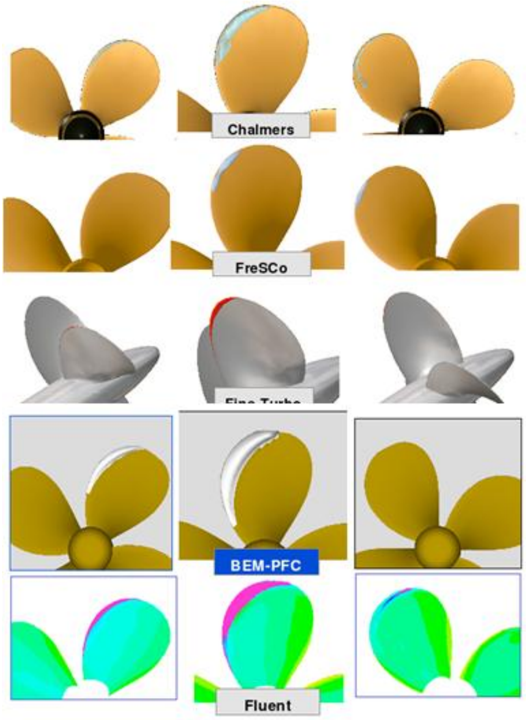


Figure 50 Unsteady, cavitating flow. Vapor fraction contour for _v=0.5 on blade suction side by RANS and LES solvers. Predicted cavity surface by BEM code. Angular positions -30o, 0o, +30o. (Salvatore, *et al*., 2009)

Simulations were carried out at model scale and the results compared with the experimental data released by INSEAN propeller E779A.

The 2.6 m long test section of the cavitation tunnel was modeled with the propeller running in it. In uniform inflow the square cross section was simplified to be a circular one, so that a quarter of the test section and one blade need to be computed by a quasi-steady approach. Multi-block structured meshes were used.

Simulations in non-uniform inflow were carried out for the whole propeller using the medium grid and the actual tunnel geometry. The axial wake simulated by the numerical model was found to be much stronger than that generated in the experiments, but the wake width was relatively well captured. As seen from Figure 51, the simulated trends of the cavity were correct though the cavitating region was under-predicted. The roll-up of the detached sheet cavity was not captured at all.

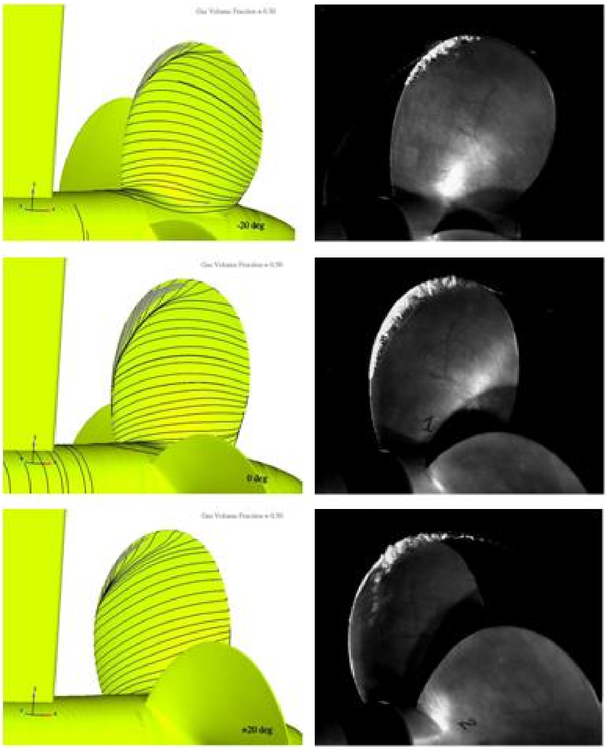


Figure 51. Comparison of simulated propeller cavitation behavior (left, vapor volume fraction = 0.5) in non-uniform inflow with photos taken from the experiments (right). From top to bottom the blade positions at -20o, 0o, and +20o are shown in propeller rotational direction (Sipilä, *et al.* 2009)

Lindau, *et al.* (2009) presented an application of the CFD code UNCLE-M developed at the Pennsylvania State University to the prediction of cavitation and associated performance for two open propellers, NSRDC P4381 and INSEAN E779A, and the JHU/NSWC waterjet pump. The multi-phase RANS solver is based on structured finite volume (FV) formulation with formal third-order spatial and second-order temporal accuracy, and employs overset grids. The k-epsilon and DES turbulence model were used in steady and unsteady computations respectively. For both propellers in open water, the computed variations of thrust and torque with the cavitation index were in fair agreement with the measured data, and the cavity extents agreed well with those observed, though the vapour volume fraction α_v equals 0.5 and 0.9 for P4381 and E779A respectively, and for E779A the angular velocity in computation is 3% lower than in experiment. The simulated variation of cavity extent for E779A in a generated wake was also shown at two cavitation indices. For the waterjet pump, the simulation results show that the thrust and torque both decrease with decreasing cavitation index when the operating condition is fixed.

The predicted thrust from single-phase computation agrees very well with experiment.

Liu, *et al.* (2008) presented a study on CFD modeling of sheet cavitation for the INSEAN E779A propeller model, using FLUENT 6.2 and the full cavitation model where the formulas given by Singhal, et al. (2002) were used to evaluate the vapor generation and condensation rates. The effect of non-condensable gas mass fraction, f_g, on predicted cavity geometry was mainly investigated. Starting from FLUENT's default, $f_g=1.5\times10^{-5}$, and successively reducing f_g to 1.5×10^{-8}, the steady sheet cavitation was simulated at J=0.77, $\sigma=1.783$. Through comparison of simulated and experimental cavity extents the authors concluded that to better predict the cavity geometry $f_g=1.5\times10^{-7}$ should be used. Based on the conclusion further simulations were carried out at J=0.71, $\sigma=1.515$ and J=0.83, $\sigma=2.016$. Taking vapor volume fraction equals 0.1 as the cavity surface, the simulated cavity extents were shown to agree well with those recorded in experiments and predicted by a BEM (Salvatore, *et al.*, 2003).

Kimura, *et al.* (2009) presented a study of performance and cavity extent prediction for two highly skewed propellers with different pitch distribution and expanded area ratio. RANS simulations were conducted using the commercial software FLUENT. A low Reynolds number two-equation turbulence model and the "full cavitation model" by Singhal, *et al.* were employed. In uniform inflow the simulated sheet cavity extent at different loading and cavitation conditions agreed fairly well with experiment, especially for the propeller less tip-unloaded. In the wake, however, the predicted cavity extent was in better agreement with experiment for the other propeller. Results of pressure fluctuations and thrust breakdown were also compared with experimental data.

Sato, *et al.* (2009) studied the applicability of cavitation prediction method for marine propellers using the commercial CFD code ANSYS CFX and the SST k-omega model. Unstructured grids were used in the small cylindrical domain surrounding and rotating with the propeller. The total number of cells was about 1.8M, resulting in a y^+ value of 1~3. The time step size was relatively large because only the low frequency phenomena were of interest in the study. The nominal axial wake was given as the inflow. The test cases include ten propellers of different number of blades, expanded area ratio, pitch ratio, and skew angle. Based on the comparison with experimental records, it was concluded that the fundamental behavior and area of sheet cavitation, as well as the 1st-order blade frequency component of pressure fluctuation were fairly well predicted. Discrepancies in cavity extent still exist when the blade moves to the 12 o'clock position, and the tip vortex cavitation was not seen from the simulation results.

Ji, *et al.* (2010) performed CFD simulation of unsteady cavitation for the full-scale Seiun-Maru highly skewed propeller using CFX. The k-omega SST turbulence model and the cavitation model proposed by Zwart, *et al.* (2004) were used. The computed cavity shapes (iso-surfaces of vapour volume fraction $\alpha_v=0.1$) on the blade at different angular positions were compared with those recorded in experiment. The change in cavity extent with time was captured well, although the cavity extent was under-predicted in general and the tip vortex cavitation existing in experiment was not captured in the simulation.

Hasuike, *et al.* (2009) presented a CFD study of propeller cavitation simulation using the commercial code SCRYU/Tetra V7. To resolve the boundary layer and tip vortex flows accurately, an adaptive mesh refinement approach was proposed and found to be effective through computations for the DTMB P4119 propeller. It was also found that the SST k-omega turbulence model could predict the open water performance accurately but underestimated the boundary layer thickness, while the k-epsilon model could predict the boundary layer thickness accurately but underestimated the propeller open water efficiency. Simulations of unsteady cavitation were carried out for the Seiun-Maru HSP by using the barotropic cavitation model and the full cavitation model respectively. The predicted cavity extents agree well with those measured when taking the contour of cavity void fraction α=0.02 as the cavity surface in the case of the barotropic model, and α=0.03 in the case of the full cavitation model. Simulations for the Seiun-Maru CP were carried out with the full cavitation model only. The cavity extents were larger than experiment when α=0.03, and close to experiment when α=0.1. For both propellers, the contours of α=0.02 at r=0.9R and r=0.95R sections were found to agree qualitatively well with measured cavity thickness distributions.

In a study of the effectiveness of cavitation control by tip load distribution, Yamasaki, *et al.* (2009) also carried out cavitation simulations using the SCRYU/Tetra V7 software. The SST k-omega model and the barotropic cavitation model were applied. The simulated unsteady cavitation patterns were shown for two propellers of which the loadings from 0.6R to the tip are constant (MP1-A) and the most unloaded (MP1-D) respectively. It was found that the predictions agreed well with experiments qualitatively, however the void fraction was very low for reasons unknown yet.

Kanemaru, *et al.* (2008, 2009a) extended the SQCM (Ando, *et al.*, 1995), to the prediction of steady and unsteady cavitation of marine propellers. Sources are distributed on the wetted surfaces, while vortex lattices on the camber surfaces according to the quasi-vortex-lattice method (Lan, 1974). Kinematic boundary condition is satisfied on both the camber and wetted surfaces to solve for the vortex and source strengths. The cavity is represented by doublets. Dynamic and kinematic boundary conditions are applied to determine, respectively, the doublet strengths and cavity thicknesses iteratively. The cavity leading edges are prescribed, and the half-open closure condition is implemented at the cavity trailing edges to determine the cavity lengths. In the method the dynamic boundary condition is strictly satisfied by including the radial velocity component in the calculation of pressure, which contributes to the improvement of predicted cavity geometry close to blade tip. The method was validated against two six-bladed propellers of different loadings in uniform inflow, and the Seiun-Maru conventional and highly skewed propellers in the ship wake. As seen in Figure 52, the steady cavity patterns predicted agree very well with those experimentally recorded, and the decreases in thrust and torque with increasing loading agree quite well with experimental data, too. In the unsteady cases, in Figure 53, it is seen that the variation in cavity extent (and thickness too, as shown in the paper) is also very well predicted. It was shown in the paper that inclusion of the radial velocity improves apparently the prediction accuracy near the tip, especially for the highly skewed propeller.

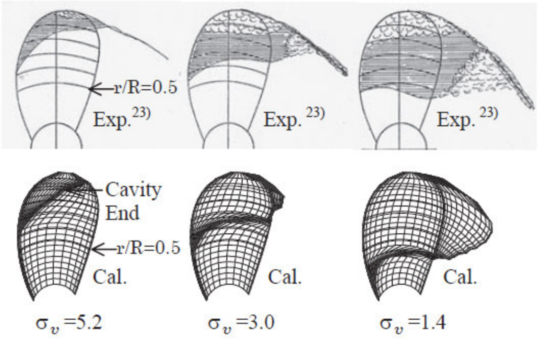


Figure 52 Comparison of recorded and computed steady cavity geometry, (P/D) 0.7R=1.264, J=0.7 (Kanemaru, et al., 2008)

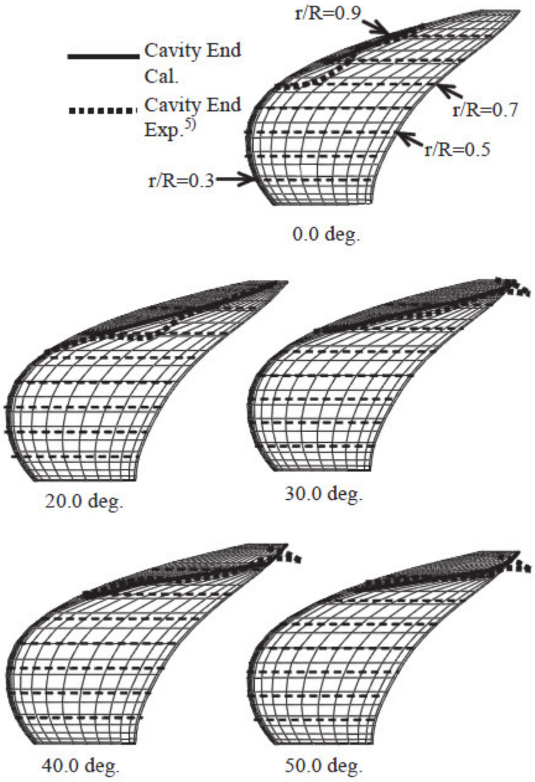


Figure 53 Comparison of recorded and computed unsteady cavity geometry, Seiun- Maru HSP, KT=0.201, _n=2.99 (Kanemaru, *et al*., 2009a)

Furthermore, Kanemaru, *et al.* (2009b) developed a method for predicting the fluctuating pressures on the hull surface. The unsteady cavity geometry is solved first and taken as a modification to the blade geometry. Then the propeller with known cavity geometry is combined with the hull, assuming the free surface as a wall. The kinematic boundary condition is implemented on the wetted surfaces of hull and blade, as well as on the cavity surfaces. The computed amplitudes of hull pressure at 1st and 2nd blade frequencies for the Seiun-Maru propellers correlate well with the measurements, though somewhat over-predicted in most cases.

To understand the evolution process and the structure of propeller trailing vortex wake, Hong and Dong (2010) performed a flow simulation for the DTMB 4119 propeller model using FLUENT. The computational domain was a circular cylinder co-axial with the propeller, in which a smaller circular cylinder surrounding the blades and the hub was cut out to form a sub-domain. The sub-domain was discretized with tetrahedral and cuboidal cells and the rest of the computational domain with structured meshes. The SST k-omega model was adopted for turbulence closure. The computed distributions of axial vorticity at a number of transverse cuts located within x/R=-0.1312~0.3281 indicate that each of the trailing and tip vortex regions consists of sub-regions. The direction of axial vorticity in each sub-region is opposite to that in its neighboring sub-region. Besides, the flux of axial vorticity keeps almost unchanged across the transverse cuts downstream of the trailing edges, while it increases within 0.85R~ 1.2R as traveling downstream, which indicates that the trailing vortices emanating from inner radii (and with the same direction as that from the tip) tend to transport (or concentrate) towards the tip vortices.

Baltazar, *et al.* (2010) presented a potential flow method for predicting the steady cavitation of propeller using a low-order BEM. The cavity source strengths are obtained by solving a reduced set of linear equations which are separated from the complete system of integral equations for both wetted and cavity surfaces, taking the cavity perturbation potential as known. The wetted and cavity perturbation potentials are obtained by solving the complete system of equations, taking the cavity source strengths as known. Iterative computations are necessary to get converged solution of the cavity geometry. The advantage of the method is that the complete system of equations, which is quite large, needs to be inverted once only, since the coefficient matrix keeps unchanged. The method was validated against the MARIN S-Propeller and the INSEAN E779A propeller, both in steady partial cavitation condition. It was shown that the cavity geometries computed by the iterative procedure were very close to those by the usual procedure which solves the wetted potential and cavity source simultaneously. For both propellers the cavity extent was over-predicted as compared with experimental observations.

Kim (2009) numerically studied the unsteady turbulent cavitation on a hydrofoil with finite span and NACA-0015 section. For the computations, a two-phase flow approach based on homogeneous mixture approximation is adopted in which liquid-vapor mixture was modeled as an inter-penetrating continuum with the phase compositions represented by volume-fraction and the inter-phase mass transfer computed using a finite-rate model derived from bubble dynamics. An implicit, finite-volume based projection algorithm was developed that couples velocity, phase compositions, and pressure. Turbulence was modeled using RANS, LES, and RANS/LES hybrid approaches (a DES variant). A suite of multiphase computational fluid dynamics (MCFD) solvers was built using OpenFOAM, an objected-oriented, open-source CFD tool-kit, being validated for steady and unsteady cavitating flows on hydrofoils and marine propulsors. LES and RANS/LES hybrid results on the hydrofoil closely reproduced the salient features of the unsteady sheet/cloud cavitation such as the breakup of sheet cavity by re-entrant jet, and the formation and collapse of cloud cavity. The lift and drag force predictions in a range of cavitation number were also found to be in good agreement with the experimental data in terms of the mean values, the root-mean-square values, and the spectral contents (Figure 54 and Figure 55).

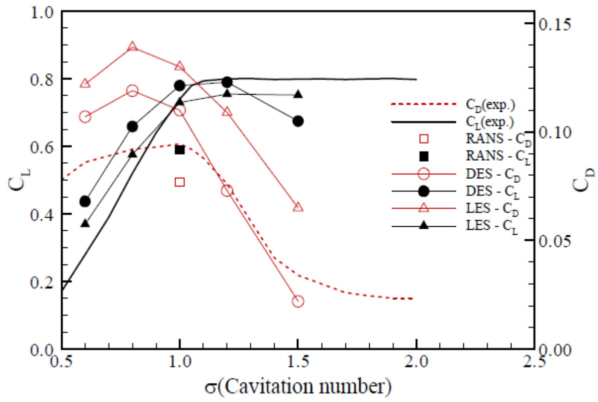


Figure 54. Time-averaged lift and drag coefficients vs. _ (Kim, 2009)

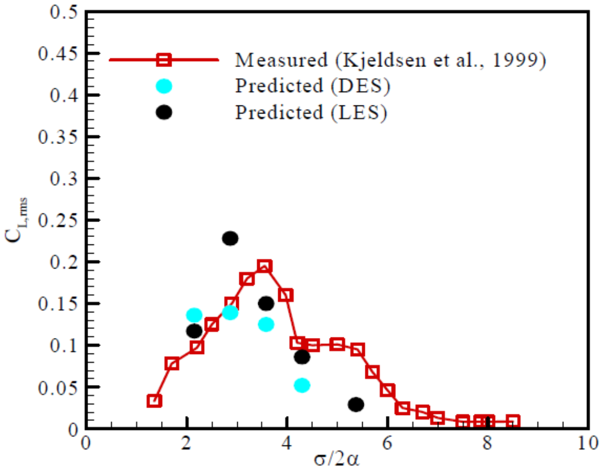


Figure 55 R.-M.-S. lift fluctuation vs. _ (Kim 2009)

Kim (2010) extended the previous work with OpenFoam to propellers and waterjets. Calculations of cavitation inception and thrust breakdown were made for the NSWCCD propeller 4381 and the ONR AxWJ-1 axial flow waterjet that was tested at NSWCCD and Johns Hopkins University.

Li, *et al*. (2009) presented a study of using a modified SST k-ω model with a multi-phase mixture flow RANS solver to predict the steady and unsteady cavitating flows around 2D and 3D hydrofoils. The cavitation was modeled by Schnerr-Sauer's cavitation model. It was found that with the modified SST k-ω model the RANS solver was able to predict the essential features like development of re-entrant jets, the pinch-off, the shedding of vortex and cloud cavities for the 2D NACA0015 foil at σ=1.0 (unsteady with shedding). For the case at σ=1.6 (stable sheet cavitation), the model predicted a high frequency sheet cavity with minor shedding at its closure. Compared with the standard SST model, the global quantities like lift, drag, and shedding frequency predicted by the modified model were closer to the experimental data, although considerable discrepancy with the experimental data was noted for the unsteady case at σ=1.0. In addition, a special type of secondary cavities, developed downstream an upstream-moving collapse cavity and termed as “vortex group cavitation”, appeared to be observable in the simulation at this condition (Figure 56).

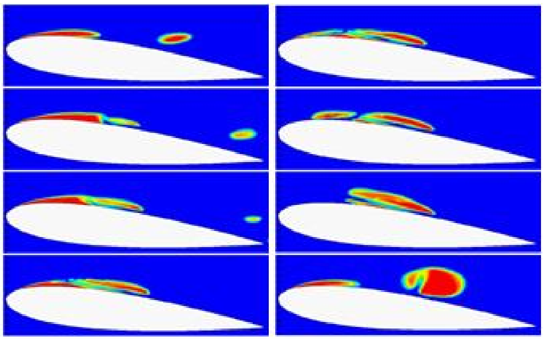


Figure 56 Time history of cavity shape expressed by vapor void fraction at _=1.0 (Li, *et al*., 2009)

Moeny, *et al*. (2008) presented a paper on generating a non-cavitating tip vortex from a foil in a water tunnel that interacted with the flows on a rudder located downstream. The interaction of the vortex with the rudder resulted in cavitation inception on the rudder and erosion damage. The flows were measured through oil-point flow visualization, erosion detection and a high speed video.

2.7 Identify the Need for R&D for Improving Methods of Model Experiments, Numerical Modelling and Full-Scale Measurements

Based on the papers published during the period of this committee and the experience of the members, the following list of topics that would be of benefit to the community were assembled.

2.7.1 Model Scale R&D Needs As PIV is becoming a routine velocity measurement technique in many towing tanks, the need for uncertainty and calibration standards has become apparent. While this may be more of a topic for the Detailed Flow Measurement Specialist Committee, it is encouraging to develop a benchmark test for PIV that was described in Section 2.2.

The differences in measurements between facilities for the simple ABB puller-pod indicates that further work is needed to accurately assess the performance of the pod in isolation before consistent results for a podded propulsion tests can be expected. The works of Glodowski, *et al.* (2009) and Savio, *et al.* (2011) show progress in this area.

The topic of wake scaling as implemented in the current ITTC procedures has been questioned by many recently and the conclusions of the specialist committee on Scaling of Wake Fields are eagerly anticipated.

The appropriate use of roughness and scaling for low Reynolds number appendages needs to be more thoroughly investigated and conclusions supported by full scale measurements. This is particularly true for pre-swirl vanes and wake influencing ducts.

2.7.2 Numerical Modeling R&D Needs

Promising advances in propulsion simulation by both in-house and commercial CFD software are made during the last three years. It seems necessary to model the true geometry of propeller and make the computational mesh sufficiently fine for both hull and propeller, in order that the propulsion factors are predicted more accurately. Meanwhile, the body-force approach remains to be an alternative that is efficient and easy-to-use for engineering purposes. There is, however, a lack of benchmark data for the validation of propulsion prediction. Further R&D work for numerical propulsion simulations are proposed as follow,

1. Study of numerical uncertainties arising from mesh resolution, turbulence modelling, and numerical discretization schemes,
2. Full scale propulsion prediction, and
3. Prediction of cavitation and fluctuating pressure for propeller operating behind the hull.

A realistic and consistent approach to simulate the effect of the correlation allowance in a full scale propulsion calculation is needed.

Since the correlation allowance captures an additional drag component, it should be added into the numerical simulation, which should result in a different full scale boundary layer at the propulsor and rudder inflow.

Various CFD codes for propeller cavitation have been widely used. Many of them can predict cavity extents qualitatively well. For further development, following issues are to be investigated.

1. Grid resolution and numerical dissipation in the vicinity of the cavity-fluid interface,
2. Turbulence and cavitation models, and
3. Modelling of a non-homogeneous inflows.

2.7.3 Full Scale Measurement R&D Needs

To support full scale trials, improved wave height and direction sensors are needed.

Methods to measure the interaction of drag-reduction techniques on propulsion performance, specifically on the wake, cavitation inception and powering performance, need to be established to ensure that source of powering differences can be appropriately characterized.

There is a need to improve the accuracy of full scale thrust dynamometers and to measure propulsor side forces and/or dynamic response.

More public domain propulsion data is needed for computational tool and scaling validation, particularly for pod applications.

If the hybrid-contra-rotating pod propulsion system becomes more common, then there will be a need for accurate measurement on the aft propeller during steady ahead and maneuvers. The loads on that propeller could lead to issues regarding bearing and seal life that will impact the pod maintenance and reliability.

3. REVIEW OF ITTC RECOMMENDED PROCEDURES RELEVANT TO PROPULSION (INCLUDING PROCEDURES FOR UNCERTAINTY ANALYSIS).

Per the instructions of the AC, the committee reviewed the following procedures:

7.5-02-03-01: Propulsion/Performance category including five sections.
7.5-02-03-02 Propulsion/Propulsor category including three sections except LDV section.

Procedure 7.5-02-03-01.1 on Propulsion Testing involved significant additions on the topic of bollard pull and will be discussed in Section 3.2.

Procedure 7.5-02-03-01.3 on Podded Propulsor tests and Extrapolation generated proposed changes to eliminate unnecessary justifications and discussions and add a nomenclature section as discussed in Section 6. The Advisory Council advised the committee to leave the procedure as is until full scale pod powering data becomes available to validate the procedure.

Procedure 7.5-02-03-01.4 on the 1978 ITTC Performance Prediction Method was updated and will be discussed in Section 5.

Procedure 7.5-02-03-02.4 on 5-hole Pitot tube wake surveys was updated with some corrections to the nomenclature used in the figures to make them consistent with the stated Nomenclature. Most significantly was the corrected labeling of the axis system shown in Figure 1 of the procedure.

Minor formatting corrections were made to Procedures 7.5-02-03-01.2 and 7.5-02-03-02.2 on uncertainty analysis.

3.1 Identify the Need for New Procedures and Outline the Purpose and Content of Them

More and more complex propulsion systems (propulsors of dual type and multiple components, such as a set of several azimuthing thrusters with nozzles, propulsors with different components each requiring a separate scale effect or, e.g, main propellers with contra-rotating podded propellers behind them) are now commonly designed and a powering prediction procedure is identified as a need. This is the topic of the questionnaire described in the next section.

Only the high speed craft propulsion testing procedure indicates that thrust loss due to cavitation needs to be modeled in propulsion predictions (7.5-02-05-02). This should be addressed in the 1978 ITTC Propulsion Prediction Method (7.5-02-03-01.4) for all types of vessels.

Develop a scaling procedure for low Reynolds number preswirl vanes, wake-influencing ducts and boss-cap fins performance.

Review the results of the 26th wake scaling specialist committee to potentially update cavitation scaling procedures so that model scale results better correlate full scale trial results.

Consider updating the 1978 (friction line, form factor, wake scaling) especially for twin propellers. Changes that should be investigated are the scaling of wake fraction for twin screw ships. As noted previously, the propeller scaling procedure should be examined to ensure it is applicable for skewed propellers and propellers with long chord tips (WCT, Kappel, CLT) based on modern CFD and full scale data.

It is important to find full scale correlation data that will allow the Predicting Powering Margins guideline (7.5-02-03-01.5) to have a roughness correction to be added to section 4.2.2.

3.2 Survey by Questionnaire

The 25th ITTC Propulsion Committee report reviewed several hybrid propulsor arrangements, aiming at propulsion efficiency gain by reducing propeller loading. The Committee concluded that the hybrid propulsion systems require specific model testing and full scale powering prediction procedures, the need for which should be further investigated.

Therefore, the present Propulsion Committee decided to prepare and circulate a dedicated questionnaire to major ITTC member organizations for their feedback in an attempt to identify potential need for developing a dedicated hybrid propulsion testing procedure. The questionnaire contained two major topics of interest. First, a group of questions regarding the types of hybrid propulsors tested, frequency of testing, testing method (variable loading/ variable speed approach), power sub-division between multiple propulsors, optimization of pod/azimuthing thruster “rudder” angle and propeller direction of rotation, availability of in-house hybrid propulsion testing procedure and willingness to share it within the ITTC community. The second group of questions concerned the derivation of propulsive factors (wake fraction, thrust deduction and relative rotative efficiency) as well as their scaling procedures. Finally, existence of full scale trial data from hybrid propulsion systems and willingness to share this information were inquired.

The questionnaire was sent out to about 55 ITTC member organizations of which 14 organizations from 7 countries responded with full or partial answers.

3.2.1 Summary of the Responses Eleven organizations (78%) conduct model tests with hybrid propulsion systems. 73% of these hybrid propulsion systems are a combination of conventional centreline (CL) propeller with a pod drive behind utilizing the CRP effect. Each 64% share the CL propeller + wing pods systems and forward/aft thruster units/pods, typical for double ended ferries. A smaller portion (36%) substitutes water jets combined with conventional propellers/pods. The average frequency of hybrid propulsor testing was reported to be about 1 to 3 projects/year at each organization.

All eleven organizations which carry out hybrid propulsor testing conduct separate propeller open water, ship model resistance and self-propulsion tests. 36% out of these treat the azimuthing thruster/pod leg as an appendage, while the rest 64% treat it as a part of the propulsor in the open water and self-propulsion tests. The hybrid propulsion testing is conducted equally with stock or final design propellers. 55% of the organizations use the variable loading propulsion test method; 18% use the variable speed approach, while the rest 27% use combined variable loading / speed methods.

In 45% of the cases, the power distribution is given by the propulsor designer; in the rest 55% the power distribution is defined experimentally based on minimum total power consumption. The majority of the organizations (55%) optimize the pod/thruster “rudder” angle experimentally by “sweeping” a discrete number of angles, while the rest 44% use RANSE CFD approach. Likewise 82% of the organizations optimize the propeller direction of rotation experimentally and the rest 18% use RANSE CFD.

It was very surprising to find out that 55% of the organizations have their own dedicated hybrid propulsor testing procedures, but the majority (82%) is NOT willing to share the procedure among ITTC community. At the same time most of the organizations (71%) expressed a need for ITTC to develop and propose a standard procedure for hybrid propulsion model testing.

With respect to propulsive factors derivation and full scale performance prediction, the majority (82%) use the thrust identity approach and 72% derive common thrust deduction coefficient for all propellers. 64% scale the effective wake and the propeller open water characteristics according to the standard ITTC-78 Powering Performance Prediction Method, the rest use some in-house developed alternative. 36% of the organizations have their own ship trial performance data, but again, 91% are not willing to share it.

3.2.3 Conclusion and Recommendation

Based on above survey, the Committee can conclude that the hybrid propulsor testing is not very frequent and is still in its pre-mature phase of development. The general conception is that institutions dealing with this subject are NOT very much willing to share their expertise in this field, but at the same time, it is feasible for the ITTC to develop and propose standardized testing procedure.

On this basis the Committee recommends that the next ITTC Propulsion Committee initiates the development of practical initial guidelines for hybrid propulsor testing, which at later stage to be further developed as a standard procedure.

The purpose of the guidelines is to ensure consistency of methodology and the acquisition of correct results for propulsion testing of hybrid propulsion systems.

The guidelines should be applicable for the following hybrid propulsion systems on basis of their highest practical importance:

- Double-ended ferries
- Triple screws with different diameter
- Centerline propeller with wing pods
- Hybrid contra-rotating propellers

3.3 Include procedure for testing of bollard pull in Recommended Procedure 7.5-02-03-01.1

Bollard pull is the static force exerted by a tug on a fixed towline. Subsequently, the bollard/trawl pull performance is typically of interest for a specialized types of ships, such as offshore supply vessels (OSV), anchor handling tugs (AHT), harbor and escort tugs, fishing trawlers, etc. The propulsion systems which are typical for the above vessel types, are single and multiple shaft lines with open and ducted propellers, azimuthing thruster units and podded drives with open or ducted propellers. Cycloidal propulsion systems could also be attributed to this cluster with the specifics that by means of measured torque, the power can be calculated but without determination of the thrust deduction.

Typically the bollard/trawl pull test is conducted as a part of the self-propulsion test, which implies that the ship model, propulsors/nozzles, measuring equipment and instrumentation is usually the same as those for the self-propulsion test. However, the bollard pull test can be distinguished from the ordinary self-propulsion test by a few major specific differences:

- The bollard pull test is conducted at zero speed of advance;
- The concepts of wake and relative rotative efficiency are no more applicable in bollard pull condition, whereas the interaction with the hull is accounted for by the familiar thrust deduction coefficient. This also implies that the propeller open water characteristics are not necessarily required for the bollard pull analysis;
- At bollard pull condition, the propeller induces very high axial and tangential velocities and actually acts as an axial pump. The flow through propeller disc is accelerated and creates a current in the towing tank, which strength is depending on the propeller loading, the tank dimensions (specifically depth and width) and the longitudinal position of the ship model/propeller relative to the tank length;
- Due to the heavy loading and induced axial and tangential velocities in the propeller slip stream, there is relatively strong interaction between the propeller and rudder, which is exhibited as internal system force and is included in the measured total bollard pull;
- At some conditions with very high loading, the propeller blades may start to ventilate due to air suction from free surface. This will significantly affect thrust and torque measurements.
- Furthermore, possible propeller cavitation and its influence on bollard pull performance cannot be modeled in a standard atmospheric pressure tank. If there is a danger of cavitation the test must be made in a pressurized tank or cavitation tunnel. The diagram in Figure 35 (Mertes and Heinke, 2008) is recommended for evaluation of the possible occurrence of cavitation.
- The trawl pull test is distinguished by the bollard pull test with its low speed of advance. This implies somewhat reduced propeller loading relative to bollard and necessity to consider the actual model resistance, corrected with the appropriate skin friction correction force.

Bollard/trawl pull tests are typically carried out with final design propulsors to verify its bollard performance. However, it could be also a common practice for the propeller designer to require bollard pull tests with stock propellers/ducts to check the hull interaction (basically the level of thrust deduction). Therefore, the proposed bollard pull procedure is equally valid for stock and final design propulsors/ducts.

Considering the major similarities with the Propulsion test procedure, the Committee decided to include the Bollard pull procedure as an integrated part (section) of the latter. Hence, the proposed Bollard pull procedure is incorporated in the updated Propulsion/Bollard Pull Test – document 7.5-02-03-01.1.

4. IDENTIFY THE PARAMETERS THAT CAUSE THE LARGEST UNCERTAINTIES IN THE RESULTS OF MODEL EXPERIMENTS, NUMERICAL MODELLING AND FULL-SCALE MEASUREMENTS RELATED TO PROPULSION

4.1 Review the largest uncertainties in the results of model experiments related to propulsion:

The potential causes of uncertainties in model test related to propulsion are the following:

- Measurements accuracy of the thrust, torque, flow velocity, carriage velocity, rate of revolution (rotational speed), water temperature and density. It is generally included in the uncertainties of the measurement sensor and the tare corrections for bearing friction (except if the sensor is included in the hub) but it should also included the statistical characteristics of the measurements signals, especially steadiness, and data processing techniques (sampling, filtering).
- Environmental conditions : blockage effect or finite depth correction (tunnel and towing tank), turbulence level, bare hub thrust correction, friction correction if there is a bearing in between the propeller and the dynamometer, surface roughness on propeller blades,
- Model geometry accuracy of the hull, the appendages and the propeller, propeller deformation when loaded in similarity to full scale, turbulence stimulation (hull, appendage, propeller)

Uncertainties in results of model propulsion experiments are composed of bias and precision errors. Precision errors are the closeness of agreement among test results obtained under prescribed conditions, which are related to repeatability of tests caused by random errors and unsteadiness. Bias errors are systematic errors that contribute to the difference between the mean of a large number of test results and an accepted reference value (Coleman and Steele, 1999).

Propeller open water test data and resistance test data are used in the analysis of the self propulsion data that all contribute experimental error sources. For open water tests and propulsion tests, there seems to be the bias is the dominant error in the total uncertainty of tests if enough number of tests is performed. For the single test case precision is the dominant error in the total uncertainties. The procedures 7.5-02-01.2 Propulsion, Propulsor Uncertainty Analysis, Example for Propulsion Tests and procedure 7.5-02-03-02.2 Propulsion, Propulsor Uncertainty Analysis, Example for Open Water Tests have both indicated this finding. Korkut, *et al.* (2005) have also shown similar results for the propulsion tests. Their results indicate that the total uncertainties in the propulsion factors are dependent on number of tests as shown in Figure 57.

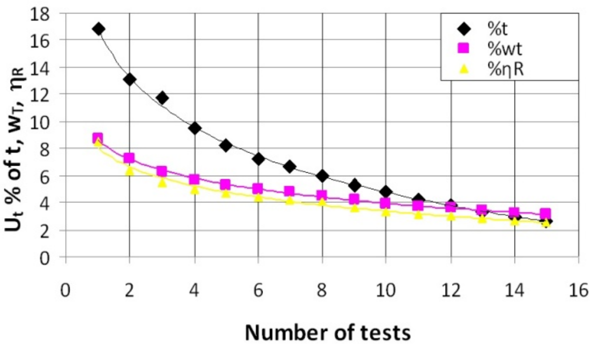


Figure 57 Total uncertainties in the propulsion factors vs number of tests (Korkut, *et al.* 2005)

The results indicate that both model thrust and external tow force are dominant parameters in the thrust deduction factor.

For the wake fraction, propeller rate of rotation, n and open water advance coefficient, J are big bias errors sources in the total bias error.

For the relative-rotative efficiency, model thrust, open water propeller results are the parameters affecting the total uncertainty. Open water propeller results are the main source of bias error in the total uncertainty.

A significant part of the bias limits are originating from embedding the uncertainties of resistance test and open water test into the propulsion test bias limits (ITTC, 2002).

English (1995) has been investigating the random uncertainty of ship propulsion experiments using a load varying method. In order to investigate the random uncertainty, two single screw ship models have been selected: Model I of a fine form moderately fast vessel and Model II of a full form vessel in ballast with stem flow separation. For this investigation, 152 ship speeds were included in the sample, requiring a minimum of 608 propeller loadings or tank runs (4 propeller loadings per vessel speed).

Propeller rotational speed and model speed were identified as the variable with the least random scatter present. The random error in the measurement of the towing force is found central to the evaluation of the random uncertainties in the other variables and derived results. English found that large models can introduce greater uncertainty of measurement, because of longitudinal model oscillation contaminating the towing force measurement. The final random uncertainties (1/2 range partial 95% confidence limits) on the ship power prediction was found to be in between ±2.02% and ±5.22% for Model I, and in between ±4.74% and ±19.17% for Model II.

Another conclusion of the work is that the non-linear regression, when variables F_D, T, Q are regressed against the propeller rotational speed n, is recommended to reduce the uncertainties.

Although the paper mainly focused in the random part of the uncertainties, the author recognizes that bias and accidental uncertainties have to consider for the uncertainty on ship power prediction. Holtrop (2001) also mentioned throughout a statistical studies of correlation of model experiments and full scale trials, that a standard deviation of 5.9% was found on power correlation coefficient. The standard deviation of the power correlation coefficient is composed of three independent contributions: uncertainty on model test, uncertainty of the extrapolation process, and uncertainty in the full-scale trials.

$$\sigma^2_{Correlation} = \sigma^2_{Model\_Test} + \sigma^2_{extrapolation} + \sigma^2_{full\_scale} \qquad (1)$$

$$5.9^2 = 3^2 + 2.5^2 + 4.5^2$$

Holtrop (2001), as English (1995), pointed out that in model experiments, the uncertainty in the results is caused primarily by the inaccuracy of the towing force measurements in captive systems or by a lack of steadiness in free running measurement systems. The correlation of the propeller rotational speed at a given power absorption has a standard deviation of 1.5%. This figure applied for fixed-pitch propeller. For controllable-pitch propellers, a much larger standard deviation applies; mainly induced by the uncertainty on the actual pitch at full scale.

4.2 Review the largest uncertainties in the results of powering predictions

An extensive sensitivity analysis has been done by Bose and Molloy (2009) and has already been reported in the final report of the 25th Specialist Committee on Powering Performance Prediction. Molloy, *et al.* (2006) have done a sensitivity study of ship powering from the ITTC database of ship and full scale trials data (Bose, *et al.* 2005). The uncertainty analysis is based on a Monte-Carlo method and the uncertainty on the inputs is computed.

All these papers are more oriented on the effect of variation in input and scaling parameters on the powering prediction. This approach does not address the problem of what kind of level of uncertainties, we should expect on the inputs, so that it does not give any actual physical uncertainty level. The varied inputs were:

Table 3 Parameters varied in Bose and Molloy (2009) sensitivity analysis

Tests	Parameters varied	% variation (standard deviation)
Resistance test	R_{TM}, V_R	1%
Propeller open water test	K_T, K_Q, J	1%
Self-propulsion test	V_M, n_M, T_M, Q_M, F_D	1%

Each input values of the three tests have been varied independently (alone) to examine the impact of individual tests (resistance, open water, self propulsion). It was found that the self-propulsion test has the most impact on the variation in power prediction and that the resistance test has the least impact. A 1% variation in all the measured values from the test data caused an average of 2.1% variation in the predicted power of all ships.

Then the frictional resistance coefficient C_f, the form factor k, the correlation allowance C_A, the wake fraction (model w_{tm} and ship w_{ts}) and the thrust deduction fraction t were varied directly by a predetermined value to evaluate the impact on the predicted power. The component of the method that has the most influence on the variation in the predicted power when varied is the frictional resistance coefficient.

Bose and Molloy (2009) are also addressing the sensitivity of the extrapolation process to the uncertainty of the inputs. The ship powering prediction, Lindgren, *et al*. (1978) used in the ITTC 1978 procedure is used in this analysis and is based on three tests: resistance test, propeller open water test (used to estimate the wake fraction of the model), self propulsion test.

It should be noted that the ITTC 1978 powering prediction method was largely derived from single screw ship data. Again like in the previous paper, the uncertainty analysis is based on a Monte-Carlo method (Molloy 2006, Molloy, *et al*. 2006) and the uncertainty on the inputs is computed.

The inputs or the parameters for which sensitivity in the ship power prediction that have been investigated are: the form factor, the friction line, correlation allowance, wake fraction scaling, thrust deduction fraction....

The uncertainty on the ship delivered power induced by the uncertainty of the friction line (based on ±1/2 the extreme differences found between the ITTC 1957 line and turbulent flat plate friction lines by Grigson (1993, 2000) and Katsui (2003)) was found to be very large: 6.58%.

Assuming that the uncertainty on the form factor can rise 100% (whether or not a form factor is used), the standard deviation on the power coefficient is 7.63%.

The prediction of the delivered ship power is found to be relatively sensitive to the wake fraction. A 10% uncertainty on the wake fraction gives a 2.16% uncertainty on the ship delivered power.

Although the correlation allowance factor is often treated as proprietary information, Molloy found that the correlation allowance with a standard deviation of 50% caused an average uncertainty of 6.14% in ship delivered power.

4.3 Review the largest uncertainties in the results of numerical modelling related to propulsion:

For numerical propulsion modelling, the sources of uncertainty are going to depend on the type of numerical modelling considered. Grid resolution, turbulence modelling, far-field boundary conditions and grid dependency of different solvers can all affect the uncertainty of solutions.

For cavitation modelling by CFD, apart from the above-mentioned sources of uncertainty for non-cavitating propeller simulation, cavitation models including parameter values therein and the choice of vapour volume fraction or void fraction values produce the largest uncertainty in the results of predicted cavity geometry.

4.4 Review the largest uncertainties in the results of full-scale measurements related to propulsion:

For full scale measurement uncertainties, it was mentioned that past ITTC committees have addressed this (Final Report and Recommendations to the 24th ITTC, from the Specialist Committee on Powering Performance Prediction, Proceedings of the 24th ITTC Vol. II, pp. 601 - 638).

An attempt to understand the magnitudes of these errors was made through analysis for a set of speed/powering trials with a series of 12 twin screws sister vessels. Each trial consists of 5 pairs of runs in opposite directions and was conducted in different environmental conditions. Hence, the whole set of trial results include errors dues to measurements, hull form production, corrections for environmental conditions.

The total error in sea trials (in % of the Power) was found to vary from 10% at 15 knots down to 8% at 24 knots

Ashcroft and Davidson (2007) present trials on four 185,000 deadweight ton VLCC tankers. The vessels type is a double hulled, twin screw, twin rudder, diesel-electric powered crude tanker designed to service between the Alaska – US West Coast trade. The full scale data were corrected to ideal conditions following the ISO15016:2000 method (Guidelines for the assessment of speed and power performance by analysis of Speed Trial Data). The major point that comes out from this extensive work is that a lot of care should be taken on the trials procedure including the measurements taken (both on the ship and environmental measurements –wind, wave, swell, current), the data collection and the pre-departure ship preparation, the test run procedure, the data quality check. In the present case, it is difficult to draw a statistics on the uncertainties of trials corrected power from a series of 4 sister ships. The authors were able to reduce the differences between the predicted power and the corrected developed power calculated from trials measurements. On the first ship trials, the corrected data showed a trial power about 18.4% greater than the predicted power. On the fourth ship trials, the corrected data showed a trial power about 5.8% greater than the predicted power. One of the major influences is related to wave and current corrections that is why specific measurement of wave / swell with a buoy and a current speed measurement (ADCP) at different water depth are found to be relevant by the authors for speed trials.

4.5 Conclusions on sources of uncertainty

Sources of uncertainty in the results of model experiments, numerical modelling and full-scale measurements related to propulsion can be classified as:
Model Scale:

- For open water tests and propulsion tests, there seems to be the bias is the dominant error in the total uncertainty of tests if enough number of tests is performed.
- Both model thrust and external tow force are dominant parameters in the thrust deduction factor.
- For the wake fraction, propeller rate of rotation, n and open water advance coefficient, J, are big bias errors in the total bias error.
- Open water propeller results are the main

source of bias error in the total uncertainty of the relative-rotative efficiency.

- A significant part of the bias limits are originating from embedding the uncertainties of resistance test and open water test into the propulsion test bias limits.
- Model accuracy, particularly for the propeller

Numerical:

- The free surface model, particularly in the transom region and near any breaking waves, can lead to uncertainties that will affect the predicted resistance and trim.
- Grid quality is a primary source of uncertainty and should be demonstrated to be sufficiently through a grid dependency study.
- The turbulence modeling used for specific aspects of a numerical propulsion solution need to have been validated against experimental data for the type of problem being solved.

Full Scale:

- The greatest source of uncertainty for full scale trials is accurate measurement of wave height and direction as well as any currents.
- For full scale trials performed on ships with controllable pitch propellers, teh accurate measurement of the ’hot’ pitch of the propeller is critical to verifying powering performance.
- The condition of the hull and propeller during a trial can be a major source of uncertainty if a cleaning has not been performed and/or a diver inspection is not available.

5. CHECK THE POSSIBILITY OF ADOPTING THE FINDINGS OF THE POWERING PERFORMANCE COMMITTEE OF 25TH ITTC FOR IMPROVING THE ITTC-78 METHOD

5.1 Findings of Powering Performance Committee of 25th ITTC

The Committee reviewed the changes that were adopted in 2008 into the procedure 7.5-02-03-01.4 (1978 ITTC Performance Prediction Method). In general the Committee agree that the former Specialist Committee made the appropriate changes. Minor corrections in some formulae’s and typographical issues with the new procedure have been found and were corrected.

5.1.1 Air Resistance A new formula for wind resistance is given in the new procedure.

$$C_{\mathrm{AAS}} = 0.5 \cdot \rho_{\mathrm{A}} \cdot V_{\mathrm{S}}^{2} \cdot C_{\mathrm{DA}} \cdot \frac{A_{\mathrm{VS}}}{S_{\mathrm{S}}} \tag{2}$$

This formula is not correct since C_{AA} must be related on water density. Only C_{DA} is related to air density.

$$R_{\mathrm{AAS}} = C_{\mathrm{DA}} \cdot 0.5 \cdot \rho_{\mathrm{A}} \cdot V_{\mathrm{S}}^{2} \cdot A_{\mathrm{VS}} \tag{3}$$

$$C_{\mathrm{AAS}} = \frac{R_{\mathrm{AA}}}{0.5 \cdot \rho_{\mathrm{S}} \cdot V_{\mathrm{S}}^{2} \cdot S_{\mathrm{S}}} \tag{4}$$

$$C_{\mathrm{AAS}} = C_{\mathrm{DA}} \cdot \frac{0.5 \cdot \rho_{\mathrm{A}} \cdot V_{\mathrm{S}}^{2} \cdot A_{\mathrm{VS}}}{0.5 \cdot \rho_{\mathrm{S}} \cdot V_{\mathrm{S}}^{2} \cdot S_{\mathrm{S}}} \tag{5}$$

$$C_{\mathrm{AAS}} = C_{\mathrm{DA}} \cdot \frac{\rho_{\mathrm{A}} \cdot A_{\mathrm{VS}}}{\rho_{\mathrm{S}} \cdot S_{\mathrm{S}}} \tag{6}$$

With $C_{\mathrm{DA}} = 0.836$ the original formula results.

$$C_{AAS} = 0.836 \cdot \frac{1.2255 \cdot A_{VS}}{1025 \cdot S_{S}} \tag{7}$$

$$C_{AAS} = 0.001 \cdot \frac{A_{VS}}{S_{S}} \tag{8}$$

Consequently the formulae for air resistance must be

$$C_{\mathrm{AAS}} = C_{\mathrm{DA}} \cdot \frac{\rho_{\mathrm{A}} \cdot A_{\mathrm{VS}}}{\rho_{\mathrm{S}} \cdot S_{\mathrm{S}}} \qquad (9)$$

The original formula includes a fixed air resistance coefficient. With the new formula it is possible to use values from wind tunnel tests available in the literature and the quality of the power prediction will be improved.

5.1.2 Appendage Drag The β-method is kept the same. This method is well proven and the most towing tanks have it in use. An alternative method is given by calculating the appendage resistance separately for each appendage using the local Reynolds numbers and form factors. If local wake fraction and form factor are known this method may be more accurate.

5.1.3 Form Factor Because of known difficulties determining the form factor and the resulting uncertainties the use of an empirical formula is proposed. That is in line with the findings of Bose and Molloy (2009).

The Performance Committee of the 13th ITTC (ITTC, 1973) published an empirical formula derived from 200 model tests. With further research work may be found a more reliable formula may be using CFD.

This Committee is not recommending any changes to the form factor section beyond what was previously accepted as part of the 25th specialist committee's procedure.

5.1.4 Further corrections in procedure 7.5-02-03-01.4. Mistakes were found in chapter 2.3 for the formulas for wake fraction and were corrected:

$$w_{TM} = 1 - \frac{J_{TM} \cdot D_{\mathrm{M}} \cdot n_{\mathrm{M}}}{V_{\mathrm{M}}} \qquad (10)$$

$$w_{\mathrm{QM}} = 1 - \frac{J_{\mathrm{QM}} \cdot D_{\mathrm{M}} \cdot n_{\mathrm{M}}}{V_{\mathrm{M}}} \qquad (11)$$

In chapter 2.4.3 the formulae for the propeller load was corrected to

$$\frac{K_T}{J^2} = \frac{S_{\mathrm{S}}}{2D_{\mathrm{S}}^2} \cdot \frac{C_{TS}}{(1-t)\cdot(1-w_{TS})^2} \qquad (12)$$

The definition that relative rotative efficiency should be equal to unity (one) in the case of a torque identity case was removed.

6. FOLLOW DEVELOPMENTS IN THE FIELD OF PODDED PROPULSION WITH A VIEW ADDRESSING THE LACK OF MODEL-SCALE AND FULL-SCALE DATA IN THE PUBLIC DOMAIN NOTED IN PROCEDURE 7.5-02-03-01.3, "PODDED PROPULSOR TESTS AND EXTRAPOLATION". INVESTIGATE THE POSSIBILITY OF IMPROVING THE PROCEDURE INCLUDING SEPARATING IT INTO LOGICAL PARTS SUCH AS RESISTANCE, PROPULSION, AND EXTRAPOLATION.

6.1 Literature survey about pod propulsion test procedures

Głodowski, *et al.* (2009) give in their presentation results of a bench mark open water test as part of the Joint Research Programme No. 4 within HydroTesting Alliance. Only the first part of benchmark tests is reported; that are results of the propeller alone. Further tests are planned with focus on aft fairing and propeller gap for propeller open water tests and end plate size / position, strut gap, hub - pod housing gap, turbulence stimulation and submergence for pod open water tests.

Richards, *et al.* (2011) gives an account of intermediate result of benchmark tests with podded propellers within the Joint Research Programme 4 of the HydroTesting Alliance. Contrary to the so-called "ABB case" for this benchmark test a standardized test procedure and setup were applied. All participating institutions have done open water propeller tests but only two institutions have finished the open water test of the pod propulsor. With standardized procedures, the model basins can deliver results which are very closed to ±2% limit.

The two available test results from the pod open water test indicate that also for those tests a good correlation between the model basins can be achieved.

Savino, *et al.* (2011) compares experimental open water characteristics of podded propellers measured by different model basins with numerical calculations carried out by two different institutions. It is concluded that experiments give more reliable results since the discrepancy between the different CFD results is larger than the spreading in experimental results between the different model basins.

Go, *et al.* (2009) describe in their paper test devices, calibration, open water and self propulsion tests as well as the extrapolation. Tests were done for a twin pod ship. They have examined the actual guideline 7.5-02-03-01.3, Podded Propulsion Test and Extrapolation. Unfortunately there is no comparison with full scale results.

Two additional papers from Islam, *et al.* (2009) and Hagesteijn and van Rijsbergen, (2009) are dealing with non stationary measurements (Section 2.2.5).

6.2 Review of pod procedure

Procedure 7.5-02-03-01.3 on Podded Propulsor Tests and Extrapolation was developed by the Specialist Committee of the 25th ITTC on Azimuthing Podded Propulsion. It is valid for podded propulsors and azimuthing thrusters working as pulling or pushing units. The test procedure consists of the following logical parts:

- Propeller open water test
- Pod unit open water test
- Self propulsion test

The propeller open water test can be treated as an option. It is necessary for special purposes i.e. for the propeller design or for determining the interaction between propeller and housing. However, it is not mandatory for a performance prediction for ships with pod propulsion. It is stated that the full scale correction can be made according to procedure 7.5-02-03-02.1, Open Water Test. Since there is a special test setup necessary and a direct connection to the pod unit open water test, the recommendation is that this section must be retained in the procedure.

The pod unit open water test is necessary for power predictions. In the procedure, the test set up and extrapolation is described very precisely.

The resistance testing of the bare hull follow Procedure 7.5-02-02-01 Resistance Tests.

There are four extrapolation procedures:

- Wake fraction scaling, according to procedure 7.5-02-03-01.4, 1978 ITTC Performance Prediction Method
- Scaling of propeller open water coefficients according to procedure 7.5-02-03-02.1, Open Water Test
- Scaling of pod housing drag as described in the procedure
- Calculation of full scale podded propulsor coefficients according to procedure 7.5-02-03-01.4, 1978 ITTC Performance Prediction Method

6.3 Recommendations regarding pod procedure

The assessment of the Propulsion Committee is that the recommended methods in the procedure represents the state of the art, contains all necessary aspects for testing and extrapolation, and are already separated into logical parts, so they should be used as it is.
In reviewing Procedure 7.5-02-03-01.3 on Podded Propulsor Tests and Extrapolation, the Committee proposes to:

- eliminate the part of resistance testing for the pod without propeller since it is not a part of the testing procedure or necessary for extrapolating the pod resistance,
- eliminate other unnecessary justifications and discussions, and
- add a nomenclature section.

If full scale measurement results become available the accuracy of the procedure has to be checked.

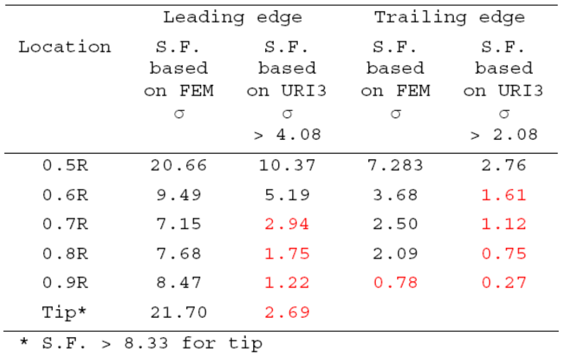
7. COMMENT ON THE IMPACT OF DEVELOPMENTS OF PROPELLERS FOR ICE GOING SHIPS IN THE VIEW OF THE INCREASING OPERATIONS IN ICE COVERED WATERS AND CHANGES IN REGULATIONS

7.1 Regulations on Ice Loading

Monitoring of current changes of unified requirements (UR) developed by International Association of Classification Societies (IACS) through website www.iacs.org.uk (in particular the chapters UR I and UR K) indicates that changes in UR for ice class propellers are related to its strength calculations and partly to pod strength. Lee, *et al.* (2007) clarifies how changes in the UR I3, been uniformly applied on ships contracted on and after 1 March 2008, influence ice class propeller design. Comparisons of blade edge strength calculations by FEM and by UR recommended simplified cantilever beam method are given with critical conclusions of UR calculations particularly for skewed propellers.

Results in Table 4 illustrate that UR I3 simplified model tends to overestimate the stress a lot compared to FEM analysis for highly skewed PC7 ice class CP propeller. 3D effect of the blade geometry should be considered in the edge strength assessment. Based on ABS current experience the conclusion is made that elastic FE stress analysis could be too conservative for some safe designs proven by their service history. In such cases, plastic FEM analysis should be adopted for correctly assessing the blade strength. Another conclusion made is as following: hydrodynamic forces acting to the blades are important and could still be the dominated source for fatigue failure in spite of the fact that compared to ice load hydrodynamic load is small.

Table 4 Safety factor SF comparison of UR and FEM results (Lee, *et al.* 2007)



	Leading edge		Trailing edge	
Location	S.F. based on FEM σ	S.F. based on URI3 σ > 4.08	S.F. based on FEM σ	S.F. based on URI3 σ > 2.08
0.5R	20.66	10.37	7.283	2.76
0.6R	9.49	5.19	3.68	1.61
0.7R	7.15	2.94	2.50	1.12
0.8R	7.68	1.75	2.09	0.75
0.9R	8.47	1.22	0.78	0.27
Tip*	21.70	2.69		

* S.F. > 8.33 for tip

Norhamo, *et al.* (2009) introduced wide review of the rules for ice class propellers with reference to recently presented IACS UR (2007) and Finnish-Swedish Ice class rules (2008). Interpretation of the rules with examples and rules background with historical references based on past 20 years' research are given. The different physics of the ice load phenomena are presented in the paper. The paper explains how ice load distributions can be derived from extreme loads predicted according to described load models. Figure 58 illustrates bent blade and corresponding FEM model of moderately skewed FP reversible propeller. Representing the appropriate ice load model, the observed damages could be computed numerically.

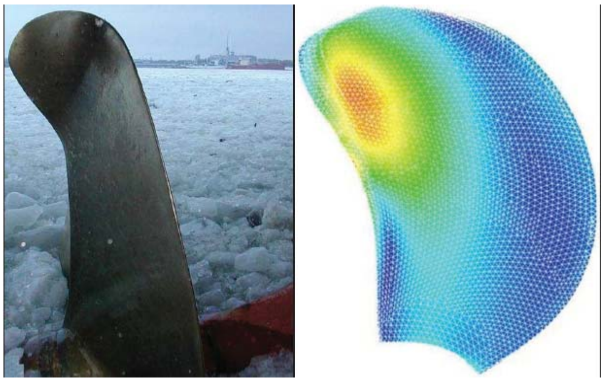


Figure 58 Bent blade and corresponding FE model (Norhamo, *et al*. 2009)

An assessment of strength of propeller blades and pitch mechanism, and components in the propulsion line requested accordingly to recently introduced rules, are discussed. Discussion on the practical consequences of designing propulsion systems to the Polar Classes is included. It is concluded that to verify compliance with the new requirements it will be necessary to increase the documentation extent to cover the whole scope. At the same time one can conclude that there are no clear indications found in new procedures requirements caused by changes in IACS UR and Finnish-Swedish Ice Class Rules. Regulatory rules of another major association - Canadian Maritime Association also have no changes indicating to needs of new procedures and model scale data.

Observation of the Finnish-Swedish Maritime Association (FMA) website indicates that two R&D themes related to ice loading are ongoing. The first topic of R&D is research on double acting ships with a CRP concept suitability for merchant icebreaking vessels trading in the Baltic Sea. The Propulsion Committee were informed by a FMA representative that this R&D project has been delayed, but nevertheless the committee concluded that this activity indicates possible requirement of hybrid propulsion system test procedures for ice going ships in future. The Committee recommends that the next ITTC Propulsion Committee initiates the development of practical initial guidelines for hybrid propulsor testing, which at later stage to be further developed as standard testing procedure. This Committee recommendation is based on review and analysis of major ITTC member organizations response to a dedicated questionnaire distributed by the Committee (see 3.1. of the present report). The second theme of FMA R&D is to develop load calculation model for Azipod-type propellers.

7.2 Ice Loading Measurement and Estimation

Hänninen, *et al.* (2007) mentioned that there are no reliable mathematical models to estimate ice loads on podded propulsion. Model scale results are difficult to scale in full scale and full scale measurement campaigns is the only way to validate ice load model for propellers and pods strength calculations. Thus from early 90's, extensive full scale measurements of ice loads on podded propulsors have been conducted on board for four vessels including MV "Norilskiy Nickel" launched in 2006 and designed to fulfill Russian Register high Arctic ice category LU7 (Figure 59).

Figure 59 Lay-out of 13 MW arctic Azipod unit installed in MV "Norilskiy Nickel" (Hänninen, *et al*. 2007)

Information on Azipod global and local ice loads and shaft line fatigue loads have been collected during the measurement projects. Based on the measurements, the most important Azipod ice loading scenarios are identified to develop optimum criteria for ice classed Azipod units.

In contradiction with Hänninen’s conclusion about full scale measurements as the way to estimate ice loads on podded propulsor propellers., Wang, *et al.* (2005a) argues that ice loads from full scale measurements have generally a high-level of uncertainty because of the non-uniform ice properties, the randomness of the interactions, uncertainties in the data acquisition systems and discrepancies among the data analysis method used. Also sampling rates are usually not high enough to capture the “real” peaks in the data. Accordingly to the paper, model tests give more precise information in terms of ice properties, the interaction conditions and the data collected. The authors utilized results of podded propeller ice interaction tests conducted in the ice tank at the National Research Council of Canada Institute for Ocean Technology (NRC/IOT) to develop a framework for analyzing ice loads on a propeller blade using probabilistic methods.

Liu, *et al.* (2008) reported on the development of a new physically based ice-hull interaction model(IHI) developed at NRC/IOT and presented the model’s numerical implementation and benchmarking based on ship model test in ice. The reference ships were the Terry Fox and CCG R-class icebreaker. Based on good correlation between the model test at NRC/IOT and sea trial results (Spencer and Jones, 2001, Jones and Lau, 2006) the data from the captive model tests were directly used for calibrating and benchmarking the model. Figure 60 shows a comparison of the measured resistance force to its predictions for level ice and pre-sawn ice. Except for the data point corresponding to 0.6 m/s ship speed the prediction are lower than measurements, but agree quite well each other with relative discrepancy smaller than 20%. This discrepancy was attributed by authors to uncertainty and non-uniformity of ice properties.

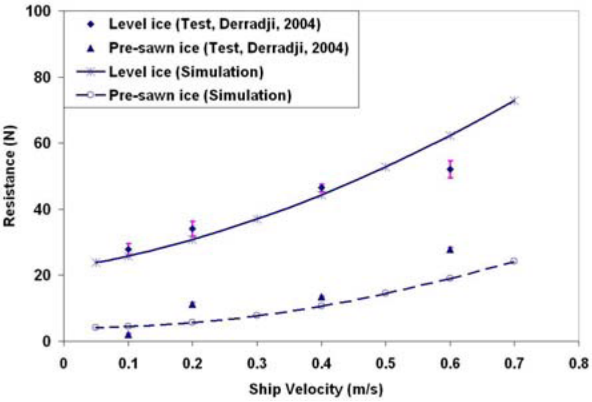


Figure 60 Comparison of measured and simulated resistance for Terry Fox Model in 40 mm - 31.5 kPa ice at test speeds ranging from 0.1 to 0.6 m/s (Liu, *et al*. 2009)

Liu, *et al*. (2008) reported on verification of a newly enhanced unsteady time-domain multiple body panel method. The numerical model against recent experimental results of Wang, *et al*. (2005b) was validated. Simulations were performed in a real unsteady case, that is, the ice piece stands still and the tractor type podded propeller moves and approaches the ice piece until collision occur to observe the transient force fluctuations in this case. Multiple-body interaction model was incorporated in an in-house propeller code PROPELLA to simulate in podded R-class propeller interacting with a sawn ice sheet under transient proximity condition. The new model was compared with previous computation and test results. Current model produced relatively reasonable results, for example, for transient shaft loading in terms of both magnitude and direction and could be used for hydrodynamic prediction under proximity condition.

Wang, *et al.* (2007a) presented the comparison of two ice load models from the JRPA#6 (Joint Research Project Arrangement between Canada and Finland) and IACS with test results of podded propulsor in IOT’s ice tank. The shaft thrust and torque coefficients from IACS model are mostly well predicted until advance coefficient reaches 0.4, but both maximal shaft thrust and torque coefficients are smaller than those from IOT's measurements when advance coefficient goes over 0.4. The model tests and numerical predictions for a podded propulsor in ice was reviewed by Wang, *et al*. (2007b). The model test of podded propulsor in two different depth of cut, two different ice conditions (pre-sawn and pack ice) and four different azimuthing angles were reported (2008). The numerical prediction of propeller performance during ice interaction was presented (2009). A panel method and an empirical formula were used for the hydrodynamic load calculations and the ice contact load calculations, respectively. This empirical model was implemented into numerical panel code.

In addition to these papers related to model scale data as the basis to develop an ice load model for podded propeller blades, another paper of Sampson, *et al*. (2009) may be included as the point of discussion. Proceeding their systematic investigations of blockage effect for the double acting concept podded propellers, the authors concluded that cavitation due to blockage is an important effect especially for double acting ships and usual experiments on ice loads in ice tanks cannot give the required information to solve the task correctly, in particular to estimate non-stationary forces. Purpose made equipment for performing these experiments in Cavitation tunnels is required as well as procedure for doing the testing.

Sampson, *et al*. (2009) address ice milling as the main scenario of propeller operation in ice. Most above mentioned Wang's works also used ice milling scenarios but some of experiments were carried out in pack ice conditions. From the above works, one is able to have insight into the mechanism for the propeller-ice interaction once the propeller blade are milling through the ice. In real life, when chucks of ice flow into the propeller, the first event is impact (or more precisely "first milling" which could have higher loads than that from consecutive milling event) and then continuous milling would occur until whole ice is passed through the propeller. Therefore the next research topic should be regarding impact load assessment at multi-direction. It will be dependent on the size, shape and inflow speed of ice blocks. After success of this topic, it could be possible to combine two work elements (impact + continuous milling) to increase the understanding and confidence with regard to performance and structural assessment of ice class propellers.

Kietzig, *et al*. (2008) reviewed different factors influencing to ice friction and their interdependence with respect to different friction regimes. The dependence of friction against ice on temperature, velocity and normal load is well understood. However, the influence of material specific parameters such as thermal conductivity, surface roughness and wettability of the slider are very difficult to isolate. Therefore, their individual impact on ice friction is not easily described. The contribution of capillary drag on ice friction, especially, lacks a profound understanding. Future research is likely to go in this direction.

Wilkman, *et al*. (2008) reported on increased ability in thick weak ice modeling at the new Aker Arctic Ice Basin with reference to actual requirements (examples of the orders) to do it. The process of making these thick/weak ice discussed. There were many other papers (from 19th IAHR 2008, OMAE 2008, OMAE 2009, OMAE 2010) related to ice properties and other more specific points of ice testing procedures briefly reviewed but not included in the present report.

7.3 Conclusion:

No any regulation rules changes were observed that may require new procedures. Friction, podded propulsor ice testing and load calculations are still challenges accordingly to the reviewed papers.

8. CONCLUSIONS

8.1 Recommendations for Next Propulsion Committee

1. Procedure Review / Update
    (a) The AC deferred the review of the procedures dealing with cavitation and propulsion in ice to the 27th ITTC period – those procedures should be reviewed.
    (b) Continue looking for full scale pod data. Get ITTC to promote joint industry project to acquire data. Refine pod procedure as full scale data becomes available, this may allow the CFD section to be removed.
    (c) The current propulsion procedure needs to address thrust loss due to cavitation in powering predictions.
    (d) Review the results of the 26th wake scaling specialist committee to potentially update cavitation scaling procedures so that model scale results better correlate full scale trial results.
    (e) Look for full scale data to improve correlation/extrapolation procedures.
    (f) Examine existing procedures and assess where CFD results can be introduced in the propulsion process to reduce or eliminate some model testing (i.e. use CFD to estimate inflow instead of wake survey for determining inflows for cavitation testing).
    (g) Consider updating the 1978 (friction line, form factor, wake scaling) especially for twin propellers
    i. Examine wake fraction scaling for twin screw ships
    ii Examine propeller scaling procedure to ensure it is applicable for skewed propellers and propellers with long chord tips (WCT, Kappel, CLT) based on modern CFD and full scale data.
    (h) Find full scale correlation data that will allow the Predicting Powering Margins guideline (7.5-02-03-01.5) to have a roughness correction to be added to section 4.2.2. Also monitor whether the EEDI may influence on margins in the future.
2. New Procedures
    (a) The Committee recommends that the next ITTC Propulsion Committee initiates the development of practical initial guidelines for hybrid propulsor testing, which at later stage to be further developed as a standard procedure.
3. Technologies to monitor
    (a) Recommend ice propulsion issues be monitored in the future. Should any significant issues develop, then a specialist committee on ice should be formed since most propulsion committee members are not knowledgeable on ice related issues.
    (b) The reduction of green house gases from ship (marine transportation) becomes more and more significant to cooperate the global warming problem. It is worthy to review the technologies (hydrodynamic issues) for enhancement of the powering performance, such as speed reduction, energy saving devices, hull form and propeller design, etc
    (c) Monitor status of CFD to perform full scale powering, resistance, cavitation and wake simulations and their correlation with full scale data.
4. Scaling for propulsors
    (a) Scaling effects of low Reynolds number preswirl vanes, wake-influencing ducts and boss-cap fins performance.

8.2 Recommendations to the Conference

The Committee recommends the Conference to adopt the changes to the following procedures:

7.5-02-03-01.1 Propulsion Test
7.5-02-03-01.2 Uncertainty Analysis Example for Propulsion Test

7.5-02-03-01.4 1978 ITTC Performance Prediction Method
7.5-02-03-02.2 Uncertainty Analysis, Example for Open Water Test
7.5-02-03-02.4 Nominal Wake Measurement by a 5-Hole Pitot Tube

9. REFERENCES

Alin, N., Chapuis, M., Fureby, C., Liefvendahl, M., Svennberg, U., and Troëng, C., 2010, “A numerical study of submarine propeller-hull interactions”, Proc., 28th Symp. on Naval Hydrodynamics, Pasadena, California, USA.

Anda, M., Iwasaki, Y., and Ebiri, K., 2010, “Development of Overlapping Propellers,” Proc. of IPS-10 International Propulsion Symposium, Okayama, Japan.

Ando, J., Maita, S., and Nakatake, K., 1995, “A simple surface panel method to predict steady marine propeller performance,” Journal of the Society of Naval Architects of Japan, Vol.178, pp.61-69.

Arroyo, P. and Hinsch, K.D., 2008, “Recent Developments of PIV towards 3D Measurements Particle Image Velocimetry”, Topics Appl. Physics, Vol. 112, pp. 127–154.

Ashcroft, F.H. and Davidson, A., 2007, “Practical aspects of conducting speed trials in the open ocean,” 10th International Symposium on Practical Design of Ships and Other Floating Structures, Houston Texas, USA.

Atkinson, C., Coudert, S., Foucaut, J.M., Stanislas, M., and Soria1, J., 2009, “Thick and Thin Volume Measurements of a Turbulent Boundary Layer using Tomographic Particle Image Velocimetry”, 8th International Symposium on Particle Image Velocimetry, Melbourne, Australia.

Baltazar, J. and Falcão de Campos, J.A.C., 2010, “An iteratively coupled solution of the cavitating flow on marine propellers using BEM,” The 9th International Conference on Hydrodynamics (ICHD’10), pp. 11-15 October, Shanghai, China.

Bazzi, T. and Benedetti, L., 2009, “Ship Model Propeller manufacturing by using RPT”, 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, Ecole Centrale de Nantes, Nantes, France.

Berchiche, N., 2008, “Numerical predictions of crashback propeller flows and loadings,” 8th International Conference on HydroDynamics, Nantes, France.

Billet, M.L., Wilson, M.B., Bailo, G.M., Gindroz, B., Lee, J.T, Bark, G., Friesh J., Hoshino, T., and Szantyr, J.A., 1996, “Report of the Cavitation Committee”, Proceedings of the 21st ITTC, Tronheim, Norway,Vol. 1, pp. 63-126.

Black, S.D. and Cusanelli, D.S., 2009, “Design and Testing of a Hybrid Shaft-Pod Propulsor for a High Speed Sealift Ship,” SNAME Propellers/Shafting 2009 Symposium, Virginia Beach, USA.

Blasques, J. P., Berggreen, C., and Andersen, P., 2008, “Hydroelastic tailoring and optimization of a composite marine propeller”, 13th European Conference on Composite Materials, Stockholm, Sweden.

Blaurock, J. and Lammers, G., 1988, “The influence of propeller skew on the velocity field and tip vortex shape in the slipstream of propellers”, SNAME Propellers ’88 Symposium, Virginia Beach, USA.

Borletau, J.P., Delfos, R., Di Felice, F., Muthanna, C., and Verhulst, M., 2009, “HTA-JRP1 : Flat plate experiment assessment of SPIV problems and accuracy in towing tank applications”, The 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, Nantes, France.

Bose, N., Insel, M., Anzböck, R., Hwangbo, S-M., Mewis, F., Steen, S., Toki, N., and Zhu, D-X., 2005, Final report and Recommendations to the 24th ITTC, The specialist Committee on Powering Performance Prediction, Proceeding of the 24th ITTC, Vol. II, Edinburg, United Kingdom.

Bose, N. and Molloy, S., 2009, "Reliability and accuracy of ship powering performance extrapolation", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Carlton, J., Radosavljevic, D., and Whitworth, S., 2009, "Rudder-propeller-hull interaction: the results of some recent research, in-service problems and their solutions", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Chang, P., Ebert, M., Young, Y., Liu, Z., Mahesh, K., Jang, H., and Shearer, M., 2008, "Propeller Forces and Structural Response due to Crashback," Proc., 27th Symp. on Naval Hydrodynamics, Seoul, Korea.

Chen, P.-F., Li, J.-Y., and Chen, L.-C., 2010, "A Study on the Technology of Energy Saving 10% (ES - 10) for Container Vessel" Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, Rio de Janeiro, Brazil.

Chesnakas, C., Donnelly, M., Jessup, S., and Lanni, F., 2008, "Testing of a Waterjet at a Range of Scales," 27th Symp. on Naval Hydrodynamics, Seoul, Korea.

Climate Change and Ships: Increasing Energy Efficiency: 2010 Proceedings, February 16-17.

Coleman, H.W. and Steele, W.G., 1999, 'Experimentation and Uncertainty Analysis for Engineers,' 2nd Edition, John Wiley & Sons, Inc., New York, NY.

Cusanelli, D., 2009, "Hydrodynamic performance comparisons between hull form and propulsor variants on a high speed sealift ship," Proc. of FAST 2009, Athens, Greece.

Di Felice, F., Pereira, F., 2008, "Developments and Applications of PIV in Naval Hydrodynamics", In Particle Image Velocimetry: New Developments and Recent Applications, Topics in Applied Physics, Vol. 112, pp. 459–485. Springer. ISBN 3540735275.

English, J.W., 1995, "Ship model propulsion experiments analysis and random uncertainty". Transaction of the Institute of Marine Engineers, Vol 197, Part 1, pp 57-81.

Felli, M., Falchi, M., and Di Felice, F., 2009, "PIV analysis on the mechanism of evolution and interaction of the propeller tip vortices with a rudder", 8th International Symposium on Particle image velocimetry PIV09, Melbourne, Victoria, Australia.

Finnish Maritime Administration, 2010, "Finnish-Swedish Ice Class Rules"

Fréchou, D., Dugué, C., Briançon-Marjollet, L., Fournier, P., Darquier, M., Descotte, L., and Merle, L., 2000, "Marine propulsor noise investigations in the hydroacoustic water tunnel GTH", 23rd Symposium on Naval Hydrodynamics, France.

Fréchou, D., Méès, L., Lebrun, D., Friedhoff, B., Lammers, G., Bugalsk, T.i, Di Felice, F., Romano, G, Verhulst, M., Hallander, J., Muthanna, C., Ireland, N., Borleteau, J.P., Ollivier, M., Delfos, R., 2009, "PIV operation in hydrodynamic facilities", Proc. 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, Nantes, France.

Fujisawa, T. and Kawamura, T., 2010, "Development of CFD method for performance prediction of contra-rotating propeller", Proc., the 5th Asia-Pacific Workshop on Marine Hydrodynamics (APHydro2010), Osaka, Japan.

Funeno, I., 2009, "Hydrodynamic optimal design of ducted azimuth thrusters", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Głodowski, R., Reichel, M., and Richards, J., 2009, "Studies on Podded Propulsors and Related Procedures – the Unified Approach", Proc. of AMT '09, Nantes, France.

Go, S., Seo, H., and Choi, G., 2009, "Study on the Powering Performance Evaluation for the Pod Propulsion Ships," Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Grigson, C.W.B., 1993, "An accurate smooth friction line for use in performance prediction", Transaction of the Royal Institution of Naval Architects, Vol. 135, pp. 149-162

Grigson, C.W.B., 2000, "A planar friction algorithm and its use in analysing hull resistance," Transaction of the Royal Institution of Naval Architect , Vol. 142, pp.76-115

Grizzi, S., Falchi, M., F.J.A. Pereira, F.J.A., Di Felice, F., 2009, "StereoPIV measurements in the wake of a piercing surface flat plate: a test benchmark for towing tank applications", 8th International Symposium on Particle Image Velocimetry - PIV09, Melbourne, Victoria, Australia.

Hagesteijn, G. and van Rijsbergen, M., 2009 "Dynamic Loads on PODs : New test set-up provides unique information," MARIN Report.

Han, K-J., Larsson, L., and Regnström, B., 2008, "A numerical study of hull/propeller/rudder interaction", Proc., 27th Symp. on Naval Hydrodynamics, Seoul, Korea.

Hasuike, N., Yamasaki, S., and Ando, J., 2009, "Numerical study on cavitation erosion risk of marine propellers operating in wake flow", Proc., 7th Intl. Symp. on Cavitation (CAV2009), Ann Arbor, Michigan, USA.

Hollenbach, U. and Reinholz, O., 2010, "Hydrodynamic Trends in Performance Optimization," Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, Rio de Janeiro, Brazil.

Holtrop, J., 1977, "A statistical analysis of performance tests results", International Shipbuilding Progress, Vol. 24, February, pp. 23-28.

Holtrop, J., 1978, "Statistical data for the extrapolation of model performance tests", International Shipbuilding Progress, Vol. 25, February, pp. 122-126.

Holtrop, J., 2001, "Extrapolation of Propulsion Tests for Ships with Appendages and Complex Propulsors", Marine technology, Vol. 38, No. 3, pp. 145-157.

Hong, F.-W. and Dong, S.-T., 2010, "Numerical simulation of the structure of propeller's tip vortex and wake," The 9th International Conference on Hydrodynamics (ICHD'10), 11-15 October, Shanghai, China.

Hänninen, S., Ojanen, M., Uuskallio, A., and Vuorio, J., 2007, "Recent Development of Podded Propulsion in Arctic Shipping", POAC-07, Dalian, China.

Hoshino, T., Jung, J., Kim, J.-H., Lee, J.-H., Han, J.-M., Park, H.-G., 2010, "Full Scale Cavitation Observation and Pressure Fluctuation Measurements by High-Speed Camera System and Correlation with Model Test", Proc. of IPS-10 International Propulsion Symposium, Okayama, Japan.

Hsin, C.-Y., Chang, K.-K., Chi, R.-C., and Chen, P.-F., 2010, "Design and Analysis of the End Plate Effect Propellers," Proc. of 28th Symposium on Naval Hydrodynamics, Pasadena, CA USA, 12-17 Sept.

IACS, 2007, "Machinery Requirements for Polar Class Ships", IACS Unified Requirements I3.

Inukai, Y., 2010 "A Development of the Electric Propulsion Vessels with Contra-Rotating Propeller" Proc. of IPS-10 International Propulsion Symposium, Okayama, Japan.

Islam, M.F., Akinturk, A., Veitch, B., and Liu, P., 2009, "Performance Characteristics of Static and Dynamic Azimuthing Podded Propulsor," Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

ITTC, 1973, Report of the Performance Committee, Appendix 4

ITTC, 1996, Report of Cavitation Committee

ITTC, 2002, Recommended Procedures and Guidelines, Propulsion, Performance Uncer-tainty Analysis, Example for Propulsion Test, 7.5-02-03-01.2.

Jang, H. and Mahesh, K., 2008, "Large Eddy Simulation of Ducted Propulsors in Crashback" Proc. Of 27th Symposium on Naval Hydrodynamics, Seoul, Korea, 5-10 Oct.

Jang, H. and Mahesh, K., 2010, "Large Eddy Simulation of Marine Propulsors in Crashback" 28th Symposium on Naval Hydrodynamics, Pasadena, CA USA, 12-17 Sept.

Jessup, S., Donnelly, M., Fry, D., Cusanelli, D., and Wilson, M., 2008, "Performance Analysis of a Four Waterjet Propulsion System for a Large Sealift Ship" 27th Symposium on Naval Hydrodynamics, Korea.

Jessup, S., Donnelly, M., McClintock, I., and Carpenter, S., 2009, "Measurements of controllable Pitch Propeller Blade loads under cavitating conditions," Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.Ji, B., Luo, X.-W., Wu, Y.-L., Liu, S.-H., Xu, H.-Y., and Oshima, A., 2010, "Numerical investigation of unsteady cavitating turbulent flow around a full scale marine propeller," The 9th International Conference on Hydrodynamics (ICHD'10), 11-15 October, Shanghai, China.

Jones, J. and Lau, M., 2006,"Propulsion and Manoeuvring Model Tests of the USCGC Healy in Ice and Correlation with Full-Scale", International Conference and Exhibition on Performance of Ships and Structures In Ice, Bauff, Alberta, Canada.

Kanemaru, T. Ando, J., 2008, "Calculation of steady cavitation on a marine propeller using a simple surface panel method," Journal of the Japan Society of Naval Architects and Ocean Engineers, Vol.7, pp.151-160.

Kanemaru, T. Ando, J., 2009a, "Calculation of unsteady cavitation on a marine propeller using a simple surface panel method," Journal of the Japan Society of Naval Architects and Ocean Engineers, Vol.9, pp.45-53.

Kanemaru, T. Ando, J., 2009b, "Calculation of propeller cavitation and pressure fluctuation on ship stern using a simple surface panel method," Journal of the Japan Society of Naval Architects and Ocean Engineers, Vol.10, pp.1-10.

Karafiath, G., Metcalf, B., and Giesbert, J., 2009, "SeaTrain for High-Capacity, High-Speed Ocean Transport," FAST 2009, Athens Greece.

Katsui, T., Asai, H., Himeno, Y, Tahara,Y., 2005, "The proposal of a new friction line".

Proceedings of the 5th Osaka Colloquium on advanced Research International Symposium on advanced research on ship viscous flow and hull form design by EFD and CFD, Osaka, Japan, pp. 76-83.

Katsui, T., Himeno, Y., and Tahara, Y., 2003, "Verification of flat plate friction coefficient at ship scale Reynolds number", Proceedings of the International Symposium on Naval Architecture and Ocean Engineering, Shanghai, China, pp. 5-1 to 5-9.

Kawamura, T. and Omori, T., 2009, "Reynolds number effect on propeller performance in open water," Journal of the Japan Society of Naval Architects and Ocean Engineers, Vol.10, pp.29-36.

Kietzig, A., Hatzikiriakos, S.G., and Englezos, P., 2008, "The Big Picture of Ice Friction", 19th IAHR Symposium on Ice, Vancouver, British Columbia, Canada.

Kim, J, Hwang, S., Lee, Y.-Y., Kim, C., Ahn, H., and Van, S.-H., 2011, "The flow field measurement around a model ship with stereo-PIV system in a towing Tank", Advanced Model Measurement Technology for the EU Maritime Industry, University of Newcastle, UK.

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

Kim, S.-E., Schroeder, S., and Jasak, H., 2010 "A MultiPhase CFD Framework for Predicting Performance of Marine Propulsors," 13th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC13), Hawaii, USA.

Kim, W.J., Van, S.H., Kim, D.H., 2001, "Measurement of flows around a modern commercial ship models", Experiments in Fluids, Vol. 31, No. 5 pp. 567-578.

Kimber, N., van der Schaaf, H., Crowther, I., 2009, "Subsea communications in a towing facility", The 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, Nantes, France.

Kimura, K., Kawamura, T., Fujii, A., Taketani, T., and Huang, Z., 2009, "Study on unsteady cavitating flow simulation around marine propeller using a RANS CFD code", Proc., 7th Intl. Symp. on Cavitation (CAV2009), Ann Arbor, Michigan, USA.

Kopke, M. and Sames, P.C., 2009, "Energy Efficiency Design Index for High Speed Craft" 10th International Conference on Fast Craft Transportaton, FAST 2009, Athens, Greece.

Korkut, E. and Danisman, D.B., 2005, "Uncertainty Analysis of Self-Propulsion Tests in Ata Nutku Ship Model Testing Laboratory," Report of Ata Nutku Ship Model Testing Laboratory, Rev.03 (In Turkish).

Krasilnikov, V. I. and Sun, J., 2008, "Verification of an unsteady RANSE method for the analysis of marine propellers for high-speed crafts", Proc., the International Conference SuperFAST'2008, St. Petersburg, Russia.

Krasilnikov, V., Sun, J., and Halse, K. H., 2009, "CFD investigation in scale effect on propellers with different magnitude of skew in turbulent flow", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

La Gala, F., Mauro, S. , Ortolani, F., Sellini, M., 2011, "A wireless underwater torque measurement system for self-pitching propellers", The 2nd International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, University of Newcastle, Newcastle, UK.

Lan, C. E., 1974, “A quasi-vortex-lattice method in thin wing theory,” Journal of Aircraft, Vol.11, No.9, pp.518-527.

Larsson, L. and Zou, L., 2010, “Gothenburg 2010 - Evaluation of resistance, sinkage and trim, self propulsion and wave pattern predictions,” Gothenburg 2010, Chalmers University of Technology, Gothenburg, Sweden.

Lavini, G., Pedone, L., and Genuzio, D. H., 2009, “Application of fully viscous CFD codes in the design of non cavitating propellers for passenger vessels”, Proc., First Intl. Symp. on Marine Propulsors (SMP’09), Trondheim, Norway.

Lee, C.-S., Choi, Y.-D., Ahn, B.-K., Jang, H.-G., and Shin, M.-S. 2010, “Performance Optimization of Marine Propellers” Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, pp. 674-680, Rio de Janeiro, Brazil.

Lee, S.-K., 2007, “Engineering Practice on Ice Propeller Strength Assessment Based on IACS Polar Ice Rule – URI3”, PRADS-2007, Houston, TX, USA.

Lee, Y. and Lin, C. 2004, “Optimized design of composite propeller”. Mechanics of Advanced Materials and Structures, Vol. 11, pp 17-30.

Li, D.-Q., Grekula, M., and Lindell, P., “A Modified SST k-ω Turbulence Model to Predict the Steady and Unsteady Sheet Cavitation on 2D and 3D Hydrofoils”, 2009 Proc. 7th Intl. Symp. on Cavitation (CAV2009), Ann Arbor, Michigan, U.S.A.

Liefvendahl, M. and Troëng, C., 2007, “Deformation and Regeneration of the computational grid for CFD with moving boundaries”, AIAA 2007-1458.

Liefvendahl, M., Felli, M., and Troëng, C., 2010, “Investigations of wake dynamics of a submarine propeller”, Proc., 28th Symp. on Naval Hydrodynamics, Pasadena, California, USA.

Lin, H. J. and Lin, J. J. 2005, “Strength evaluation of a composite marine propeller” Journal of Reinforced Plastics and Composites, Vol. 24, No. 17, pp 1759-1769.

Lindau, J. W., Moody, W. L., Kinzel, M. P., Dreyer, J. J., Kunz, R. F., and Paterson, E. G., 2009, “Computation of cavitating flow through marine propulsors”, Proc., First Intl. Symp. on Marine Propulsors (SMP’09), Trondheim, Norway.

Liu, D., Hong, F., Zhao, F., Zhang, Z., 2008, “The CFD analysis of propeller sheet cavitation,” 8th International Conference on Hydrodynamics (ICHD’08), Nantes, France.

Liu, J., Lau, M., and Williams, F.M., 2008, “Numerical Implementation and Benchmark of Ice-Hull Interaction Model for Ship Manoeuvring Simulations”, 19th IAHR Symposium on Ice “Using New Technology to Understand Water-Ice Interaction”, Vancouver, British Columbia, Canada.

Liu, P., Akinturk, A., He, M., Islam, M.F., and Veitch, B., 2008, “Hydrodynamic Performance Evaluation of an Ice Class Podded Propeller under Ice Interaction”, Proceedings of the 27th International Conference on Offshore Mechanics and Arctic Engineering, Estoril, Portugal.

Liu, X. L., 2009, “A potential based panel method for prediction of steady and unsteady performances of contra-rotating propellers”, Proc., First Intl. Symp. on Marine Propulsors (SMP’09), Trondheim, Norway.

Longo, J. and Stern, F., 2005 "Uncertainty Assessment for Towing Tank Tests With Example for Surface Combatant DTMB Model 5415", Journal of Ship Research, Vol. 49, No. 1, March 2005, pp. 55-68.

Ma, C., Du, D., Qian, Z.-F., Yang, C.-J., and

Cheng, K., 2008, "A potential method for steady and unsteady hydrodynamic performance computation of podded propulsors," 8th International Conference on Hydrodynamics (ICHD'08), Nantes, France.

Mées, L., Lebrun, D., Allano, D., Françoise Walle, F., Lecoffre, Y., Boucheron, R., and Fréchou, D., 2010, "Development of interferometric techniques for nucleisize measurement in cavitation tunnel", 28th Symposium on Naval Hydrodynamics, Pasadena, California.

Mées, L., Lebrun, D., Fréchou, D., and Boucheron, R., 2011, "Interferometric laser imaging technique applied to nuclei size measurements in cavitation Tunnel", Advanced Model Measurement Technology for the EU Maritime Industry AMT11, University of Newcastle, UK.

Mertes, P. and Heinke, H.-J., 2008, "Aspects of the Design Procedure for Propellers Providing Maximum Bollard Pull," ITS 2008, Singapore.

Mewis, F., 2009, "A Novel Power-Saving Device for Full-Form Vessels," Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Miller, M. L., 1976, "Experimental determination of unsteady forces on counterrotating propellers in uniform flow", DTNSRDC Report SPD-659-01.

Miller, R., Kim, S-E., and Lee, S.W., 2010, "A Computational Framework for Fluid-Structure Interaction on Flexible Propellers Involving Large Deformation" Proc. 28th Symp. On Naval Hydrodynamics, Berkley, USA.

Minchev, A., Nielsen, J. R. and Lundgren, E., 2009, "Ducted propeller design and verification for contemporary offshore support vessels", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Miorini, R.L., Wu, H., and Katz, J., 2010, "The internal structure of the tip vortex within the rotor of an axial waterjet pump", Proceedings of ASME Turbo Expo 2010: Power for Land, Sea and Air GT2010, June 14-18, Glasgow, UK.

Moeny, M., Weldon, M., Stinebring, D., Straka, W., and Pierzga, M, 2008, "Cavitation Damage from an Induced Secondary Vortex" Proceedings of the 2008 ASME Fluid Engineering Conference, Jacksonville, Florida, USA.

Molloy, S., Bose, N., and Reilly, D., 2006, "A sensitivity study of ship powering prediction using Monte Carlo simulation", Proceedings of the International Conference in Marine Hydrodynamics, Naval Science and Technological Laboratory, Visakhapathnam, India, pp. 377-388.

Müller, S.-B., Abdel-Maksoud, M., and Hilbert, G., 2009, "Scale effects on propellers for large container vessels", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Muscari, R., Felli M., and Di Mascio, A., 2010, "Numerical and experimental analysis of the flow around a propeller behind a fully appended hull" 28th Symposium on Naval Hydrodynamics, Pasadena, CA USA.

Nagaya, S., Ochi, F., Fukui, Y., Omori, T., and Inukai, Y., 2011, "Stereo PIV measurements of flow around energy saving device and twin-skeg stern ship in towing tank", Advanced Model Measurement Technology for the EU Maritime Industry, University of Newcastle, UK.

Nakaie, Y., Ohta, Y., and Hishida, K., 2009, "Flow Measurement around a Cycloidal Propeller", 8th International Symposium on Particle Image Velocimetry - PIV09, Melbourne, Victoria, Australia.

Norhamo, L., Bakken, G.M., Deinboll, O., and

Iseskar, J.J., 2009, "Challenges Related to Propulsor-Ice Interaction in Arctic Waters", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Oh, S., Jang, J., Rhyu, S., Hoshino, T., and Seo, J., 2010, "Some Applications on Extrapolation Method of Rudder Bulb Performance as an Energy-Saving Device" Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, Rio de Janeiro, Brazil, pp. 383-390.

Otsubo, S., 2010, "Designing a Global and Effective Regulatory System to Reduce the CO2 Emission from International Shipping – Challenge for Maritime Industries" Proc. of IPS-10 International Propulsion Symposium, Okayama, Japan.

Pereira, F., Aloisio, G., Bretschneider, H., Johannsen, C., Grekula, M., Bark, G., van de Bunt, E., Veisluis, M., and Gelderblom, E., 2009, "Methods and procedures for high-speed video recording and analysis of pressure fluctuations in propeller cavitation," 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, Ecole Centrale de Nantes, Nantes, France.

Peri, C., Campana, E., Tahara, Y., Takai, T., Kandasamy, M., and Stern, F., 2010, "New developments in Simulation-Based Design with application to High Speed Waterjet Ship Design" Proc. Of 28th Symposium on Naval Hydrodynamics, Pasadena, CA USA.

Richards, J., Glodowski, R., Koushan, K., Alterskjaer, S.A., Atlar, M., and Saisto, I., 2011, "Comparison of experimental predictions of open water performance characteristics of a podded propeller", Proc., Advanced Model Measurement Technology for EU Maritime Industry – AMT'11, Newcastle upon Tyne, U.K.

Ruiz, L., Whittlesey, R., and Dabiri, J., 2010, "Vortex-enhanced propulsion", Journal of Fluid Mechanics, Volume 668, pp 5-32.

Salvatore, F., Streckwall, H., and van Terwisga, T., 2009, "Propeller cavitation modelling by CFD - Results from the VIRTUE 2008 Rome workshop", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Salvatore, F., Testa, C., and Greco, L., 2003, "A viscous/inviscid coupled formulation for unsteady sheet cavitation modeling of marine propellers," Fifth International Symposium on Cavitation (CAV2003), Osaka, Japan.

Sampson, R., Atlar, M., and Sasaki, N., 2009, "Propulsor Ice Interaction-Effect of blockage proximity", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Sánchez-Caja, A., Sipilä, T. P., and Pylkkänen, J. V., 2009, "Simulation of viscous flow around a ducted propeller with rudder using different RANS-based approaches", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

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., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Sato, K., Oshima, A., Egashira, H., and Takano, S., 2009, "Numerical prediction of cavitation and pressure fluctuation around marine propeller", Proc., 7th Intl. Symp. on Cavitation (CAV2009), Ann Arbor, Michigan, USA.

Savio, L., Viviam, M. , Ferrando, M., and Carrera, G, 2009a "Propeller cavitation 3D reconstruction through stereo-vision algorithms," 1st International Conference on Advanced Model Measurement Technology for the EU Maritime Industry, Ecole Centrale de Nantes, Nantes, France.

Savio, L., Viviani, M., Conti, F., and Ferrando, M., 2009b "Application of computer vision techniques to measure cavitation bubble volume and cavitating tip vortex diameter.", 7th International Symposium on Cavitation (CAV2009), August 17-22, Ann Arbor, Michigan, USA.

Savio, L., Sileo, L., Steen, S., Glodowski, R., Hoffmann, P., and Kraskowski, M., 2011, "Comparison of experimental and numerical predictions of open water performance characteristics of a podded propeller", Proc., Advanced Model Measurement Technology for EU Maritime Industry – AMT'11, Newcastle upon Tyne, U.K.

Schmiechen, M., 2009, "50 years Rational theory of propulsion", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Shimamoto, H., Takeda, A., and Miyake, S., 2010, "Tandem hybrid CRP system," Proc. of IPS-10 International Propulsion Symposium, Okayama, Japan.

Sileo, L. and Steen, S., 2009, "Numerical investigation of the interaction between a stern tunnel thruster and two ducted main propellers", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Sileo, L. and Steen, S., 2010, "Numerical Investigation of the Interaction Effects Between a Stern Tunnel Thruster and Twin Ducted Main Propellers" Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, Rio de Janeiro, Brazil, pp. 133-143.

Singhal, A.K., Athavale, M.M., Li, H., and Jiang, Y., 2002, "Mathematical basis and validation of the full cavitation model," Journal of Fluids Engineering, Vol.124, No.3, pp.617-624.

Sipilä, T., Siikonen, T., Saisto, I., Martio, J., and Reksoprodjo, H., 2009, "Cavitating propeller flows predicted by RANS solver with structured grid and small Reynolds number turbulence model approach", Proc., 7th Intl. Symp. on Cavitation (CAV2009), Ann Arbor, Michigan, USA.

Soloff, S, Adrian, R., and Liu, Z., 1997, "Distortion compensation for generalized stereoscopic particle image velocimetry.", Measurement Science Technology Vol. 8, pp.:1441–1454.

Spencer, D. and Jones, J., 2001,"Model-Scale/ Full-Scale Correction in Open water and Ice for Canadian Coast Guard 'R-Class' Icebreakers", Journal of Ship Research, vol.45, pp.249-261.

Streckwall, H. and Salvatore, F., 2008, "Results from the Wageningen 2007 Workshop on propeller open water calculations including cavitation", Marine CFD, Royal Institution of Naval Architects.

Taketani, T., Kimura, K., Ishii, N., Matsuura, M., and Tamura, Y., 2009, "Advanced design of a ducted propeller with high bollard pull performance", Proc., First Intl. Symp. on Marine Propulsors (SMP'09), Trondheim, Norway.

Thomas, L., Grulier, V., Braud, P., Tremblais, B., Germaneau, A., Dupré, J.C., Doumalin P., and David, L., 2009, "Comparison of algebraic reconstruction and optical scanning for tomographic PIV", 8th International symposium on Particle Image Velocimetry, Melbourne, Australia.

Tillmanns, M., 2009, "Submarine Propeller made of composite materials", Undersea Defense Technology 2009, Cannes France.

Ueda, N. and Numaguchi, H., 2006, "The First Hybrid CRP-POD Driven Fast ROPAX Ferry in the World", www.mesj.or.jp/mesj_e/english/pub/ap_pap ers/pdf/2006AP4.pdf, Translated from Journal JIME Vol.40, No.2.

Van der Ploeg, A., Raven, H., “2010 CFD-Based Optimization for Minimal Power and Wake Field Quality” Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, Rio de Janeiro, Brazil, pp.92-101.

Vartdal, B.J., Tormod, G., and Ingvar, A., 2009, “Lateral Propeller Forces and their Effects on Shaft Bearings,” Proc., First Intl. Symp. on Marine Propulsors (SMP’09), Trondheim, Norway.

Veikonheimo, T., 2006, "Solving problems of propeller model scaling", The Naval Architect, July/August 2006, , pp. 51-53.

Verhulst, M. and Hooijmans, P., 2010, “Quasi-Steady Model Propulsion Measurements for Complex Propulsion Systems”, Proc. of 11th International Symposium on Practical Design of Ships and Other Floating Structures, Rio de Janeiro, Brazil, pp.695-702.

Wang, J., Akinturk, A. ,Jones, S.J., and Bose, N., 2005, “Model Podded Propeller-Ice Interaction in Extreme Conditions using a Probabilistic Model” , 7th Canadian Marine Hydromechanics and Structure Conference, Halifax, Nova Scotia, Canada.

Wang, J., Akinturk, A., Jones, S.J., and Bose, N., 2005, “Ice Loads on a Model Podded Propeller Blade in Milling Conditions”, Proceedings of the 24th International Conference on Offshore Mechanics and Arctic Engineering, Halkidiki, Greece,

Wang, J., Akinturk, A., Jones, S., Bose, N., Kim, M., and Cun, H., 2007, “Ice Loads Acting on a Model Podded Propeller Blade”, Journal of Offshore Mechanics and Arctic Engineering, 129(3) pp. 236-244.

Wang, J., Akinturk, A.,Jones, S., and Bose, N., 2007, “An Overview of Model Tests and Numerical Predictions for Propeller-Ice Interaction” , 8th Canadian Marine Hydromechanics and Structure Conference, St. John’s, NL.

Wang, J., Akinturk, A., Bose, N., Jones, S.J., Song, Y., Chun, H. and Kim, M., 2008, ”Experimental Study on a Model Azimuthing Podded Propulsor in Ice”, Journal of Marine Science and Technology, 13(3), IR-2007-53, pp.244-255.

Wang, J., Akinturk, A., and Bose, N., 2009, ”Numerical Prediction of Propeller Performance during Propeller-Ice Interaction”, Marine Technology and SNAME News, Vol.46, No.3, pp.123-139.

Wilkman, G., Forsen, A.-C., and Mattson, T., 2008, “Latest Development in Ice Model Testing”, 19th IAHR Symposium on Ice Using New Technology to Understand Water-Ice Interaction, Vancouver, British Columbia, Canada.

Wu, H., Miorini, R.L., and Katz, J., 2009, “Tip Leakage Vortex Structure and Turbulence in the Meridional Plane of an Axial Pump”, 8th International Symposium on Particle Image Velocimetry - PIV09, Melbourne, Victoria, Australia.

Yamasaki, S., Okazaki, A., Hasuike, N., Kawanami, Y., and Ukon, Y., 2009, “Numerical and experimental investigation into cavitation of propellers having blades designed by various load distributions near the blade tips”, Proc., First Intl. Symp. on Marine Propulsors (SMP’09), Trondheim, Norway.

Yamatogi, T, Murayama, H. K. Uzawa, K., Mishima, T., and Ishihara, Y., 2010, “Study on Composite Marine Propellers”, Proc. of IPS-10 International Propulsion Symposium, Okayama, Japan.

Young, Y.L., 2010, “Analysis and Design of Advanced Marine Propulsors” Proc. of the

ASME 29th International Conference on Ocean, Offshore and Arctic Engineering, June 6-11, Shanghai, China

Young, Y.L., Baker J.W., and Motley M.R., 2010, "Reliability-based design and optimization of adaptive marine structures", Composite Structures, Volume 92, Issue 2, January 2010.

Young, Y.L., Liu, Z., and Motley, M.R., 2008, "Influence of Material Anisotropy on the Hydroelastic Behaviors of Composite Marine Propellers" Proc. 27th Symp. On Naval Hydrodyanmics, Seoul, Korea.

Zhang, Z.-R., Hong, F.-W., Krasilnikov, V., Tang, D.-H., "Progress in analysis of viscous flow around podded propulsor," 8th International Conference on Hydrodynamics (ICHD'08), Nantes, France.