The Resistance Committee

Final Report and Recommendations to the 24th ITTC

1. INTRODUCTION

1.1 Membership

The 24th ITTC Resistance Committee consisted of:

- Dr. Emilio F. Campana (Chairman). Instituto Nazionale per Studi ed Esperienze di Architettura Navale, Italy.
- Dr. Yoshiaki Kodama (Secretary). National Maritime Research Institute, Japan.
- Mr. Peter Bull. QinetiQ, United Kingdom.
- Dr. A. H. (Sandy) Day. Universities of Glasgow and Strathclyde, United Kingdom.
- Dr. Joseph Gorski. Carderock Division, Naval Surface Warfare Center, U.S.A.
- Dr. Yuzo Kusaka. Akishima Laboratories (Mistui Zosen) Inc., Japan.
- Prof. Seung-Hee Lee. Inha University, Korea.
- Dr. Juha Schweighofer. Helsinki University of Technology, Finland
- Dr. Jesus Valle. Canal de Experiencias Hidrodinamicas de El Pardo, Spain.

1.2 Meetings

The Committee met 5 times:

- Rome, Italy, December 2002.
- Busan, Korea, September 2003.
- Helsinki, Finland, August 2004.
- Madrid, Spain, December 2004.
- Osaka, Japan, March 2005.

1.3 Tasks

Below we list the tasks carried out by the 24th Resistance Committee (RC), based on the recommendations given by the 23rd ITTC.

Task 1: Continue review of trends in experimental fluid dynamics - EFD. Monitor developments in measurement methods especially in optical techniques for measuring flow velocity, pressure, body motion, and techniques for wave profiles, especially near bow and stern.

Task 2: Review Uncertainty Analysis in experimental fluid dynamics and verify how it is utilised to improve accuracy of not only raw data but also derived quantities.

Task 3: Review the development and identify the need for research in the computation at full scale, free surface treatment, unsteady flows, design methods and optimization, and accurate modelling of turbulence. Validation by reliable data from experiments.

Task 4: Develop ITTC recommended procedures including uncertainty analysis for additional towing tank measurements as needed (e.g., nominal wake). Develop benchmark tests for Gothenburg 2000 Workshop test cases (standard tanker, container, and combatant standard ship models/propellers) between ITTC member institutes for comparative measurements and uncertainty analysis for identifying facility biases and improving the insight on the facility operation. Improve recommendations with regard to scale effects for model size and turbulence stimulation.

Task 5: Continue to monitor new developments in Verification and Validation methodology and procedures and further update and improve ITTC recommended Procedure 7.5-03-01-01 "Uncertainty Analysis in CFD, Uncertainty Assessment Methodology and Procedures". Prepare ITTC recommended Procedure 7.5-03-02-01 "Uncertainty Analysis in CFD, Examples for Resistance and Flow," based on a collective example of as many participants as possible following ITTC recommended Procedure 7.5-03-01-01 "Uncertainty Analysis in CFD, Uncertainty Assessment Methodology and Procedures" for Gothenburg 2000 Workshop test cases. Update benchmark database to validate numerical procedures, including recommendations for archiving, distribution, and use of data.

Task 6: Far Field Waves and Wash. Continue to monitor development of wash prediction techniques, in particular for trans-critical and supercritical regimes. Propose guidelines when enough experience has been collected.

Task 7: Identify developments in modelling of relevance to resistance. In particular review research and development in modelling and turbulence stimulation and provide recommendations for scaling and extrapolation. Monitor and follow the development of new experimental techniques and extrapolation methods.

Task 8: Review the state-of-the-art, comment on the potential impact of new developments on the ITTC in ship concepts, design methods and de-sign optimization and identify the need for research and development.

Task 9: Provide recommendations for scaling and extrapolation.

2. TRENDS IN EXPERIMENTAL FLUID DYNAMICS

2.1 Introduction

Experimental Fluid Dynamics (EFD) plays an essential role in the ITTC community. It has started by measuring only macroscopic forces, but has gradually been extended to detailed measurement of distributed quantities. This tendency has been enhanced by developments in CFD, which needs detailed data for validation, and EFD and CFD are becoming more closely linked for better hull form design.

This section reviews the development of EFD carried out in towing tanks and other related test facilities. Some kinds of experiments have increased in importance, and there is steady progress in measurement techniques. Developments in experimental facilities are reported, based on a questionnaire distributed to the ITTC organisations to monitor their development since June 2001.

2.2 Experiments of increasing importance

Comprehensive Measurement for CFD Validation. The development of CFD requires comprehensive experimental data in large quantities to be compared with computations for CFD validation. This type of experiment has increased in importance and there has been steady accumulation of experimental data.

Kim et al. (2001) carried out a comprehensive experiment using three modern commercial hull forms: one container ship KCS (KRISO container ship) and two very large crude-oil carriers (VLCCs) with the same forebodies and slightly different aftbodies (KVLCC and KVLCC2). They measured resistance, wave pattern, local mean velocity distribution, and carried out uncertainty analysis. The results were used for CFD validation in CFDWS2000 and CFDWS2005.

Gui et al. (2001, 2002) carried out a comprehensive experiment using a DTMB5512 surface combatant hull form, a fully appended 3.048m-long geosim of 5.72m-long DTMB5415, in regular head waves, and measured forces, moment, and wave pattern. The results were used in CFDWS2005. Olivieri et al. (2003) gave an overview of overlapping experiments on DTMB5415 carried out at INSEAN, IIHR, and DTMB.

High Reynolds Number Experiment. There is a systematic trend toward a larger scale, i.e. toward a higher Reynolds number, in experiments carried out from the resistance point of view. Bourgoyne et al. (2003) measured flows around a solid-bronze wing section of modified NACA16 (2.1m chord, 3.0m span, 17cm maximum thickness) in the U.S. Navy's Large Cavitation Channel of 3.05m × 3.05m × 13m test section, at flow speeds from 0.25 to 18.3m/s, corresponding to the a chord-based Reynolds number of more than 5×10^7. Castillo et al. (2004) measured a flat plate boundary layer using an X-probe hotwire in a wind tunnel with and without roughness up to 30m from the test section inlet at wind speeds of 10 and 20 m/s, thus reaching momentum thickness Reynolds number R_θ =120,000. They found that the velocity deficit profile collapse when they are normalised by the Zagarola/Smits scaling $U_\infty \frac{\delta^*}{\delta}$.

Full-Scale Experiment. Experimental data at full-scale is indispensable for the development of methods to predict the full-scale performance of ships.

Recently, extensive studies on drag reduction using micro-bubbles have been carried out. In this approach, air bubbles are injected into the boundary layer of ships. A full-scale experiment on the method was carried out recently. Ikemoto et al. (2002) measured local void ratios by taking photographs of air bubbles across the boundary layer of a full scale ship. Kodama et al. (2002) and Nagamatsu et al. (2002) described in detail a full scale micro-bubble experiment carried out using a 116m-long ship. In the experiment, propeller thrust and torque, injected air rate, local skin friction, local void ratio, bubble trajectories, and stern vibration were measured. It was found that bubbles entrained into the working propeller adversely affected the propeller performance but, by carefully choosing the point of air injection, bubble entrainment was avoided and a 3% drag reduction was obtained.

2.3 New Developments in Measurement Techniques

There has been steady progress in measurement techniques for experiments in towing tanks, water tunnels, and wind tunnels. The progress has been realised by adopting newly developed hardware, particularly MEMS technology, and software, and ever-increasing computer power.

Velocity Measurement. *PIV/PTV:* Since this technique of measuring flow velocities by monitoring movement of tracer particles has become very popular and there have been numerous papers published, only those related to marine hydrodynamics and towing tanks are referred.

Lee et al. (2002) measured the wake of a 5-bladed marine propeller of 54mm diameter for a 3600TEU container vessel using adaptive hybrid 2-frame PTV at a free-stream velocity of 0.325m/s, and clarified that slipstream contraction occurs up to x=0.5D, and that unstable oscillation occurs afterwards due to tip vortex separation from the wake sheet.

Fu et al. (2002) measured the cross-flow wake of a turning submarine model (ONR Body-1), a 5.18m-long axi-symmetric body with a sail and four identical stern appendages, using a submersible PIV system in a rotating arm basin. They found complex interactions of vortical structures from the hull, the sail, and various control surfaces.

3D PIV: Calcagno et al. (2002), using 3D PIV, measured the wake of a 5-bladed, D=0.22m MAU propeller behind a 6.1m long Series 60 C_b=0.60 model at flow speed of 1.22m/s and obtained the 3D velocity, vortical structure and turbulence intensity behind the propeller.

5-Hole Probes: In order to assess feasibility of MEMS-based, miniature-sized, high-frequency 5-hole probes, Babinsky et al. (2001) bored 5 pressure holes on one planar end of a circular cylinder of 25mm diameter, and measured the directional sensitivity of the pressures.

Ultrasound: Pulsed UDV (ultrasound Doppler velocimetry) can be used to measure velocity along the path of sound propagation. Nowack (2002) used the method to measure the velocity profile in a circular tube of 35.4mm inner diameter, and fitted the data to obtain wall shear stress, whose overall error was estimated to be $\pm 8.4\%$.

Flow Visualization. Furey and Fu (2002) used a laser sheet to illuminate the bow wave of a planing ship model, and used the images to detect the free surface interface. The result agreed well with conventional finger-probe data only when the surface was not largely dynamic, i.e. when there were not a large number of discrete scatterers.

Skin Friction Measurement. Tachie et al. (2001) collected data for open channel boundary layers and gave a formula for skin friction coefficient as a function of the momentum thickness Reynolds number R_θ in the range $150 < R_\theta < 15{,}000$.

Frictional Resistance and Roughness. Schultz (2002) towed a 1.5m-long flat plate immersed vertically in a towing tank at speeds between 2.0 and 3.8m/s. The plate three kinds of surface, i.e. as-sprayed paint, smoothed by sanding, and roughened with sandpaper as fine as 600 grit. The as-sprayed paint surface had 7.3% larger frictional resistance than the surface smoothed by sanding, whilst the sandpaper-roughened surface had significantly larger frictional drag that the polished surface. The results showed that the centreline average height R_a is sufficient to explain the variance in the roughness function ΔU^+ in this Reynolds number range.

Tanaka et al. (2003) devised a drum-type device to measure skin friction. They made a circular cylinder of 0.3m diameter and 0.3m height from vinyl chloride and measured torque, converted to skin friction by first subtracting idle torque and then divided by the form factor $1 + k = 1.96$, which takes into account the skin friction at two sides of the cylinder and others. Surfaces of various degrees of roughness were tested.

2.4 Developments in Experimental Facilities of the ITTC Organizations

A questionnaire on new test facilities and model manufacturing machines was distributed to the ITTC organisations to monitor their development since June 2001. Summary of the 30 replies is shown below.

New Experimental Facilities.

1. Shallow Water Basin
   Australian Maritime College (Australia)
   L: 35.0m, W: 12.0m, D: 0 to 1.0m.
   Towing carriage: Auxiliary carriage, Speed range: 0 to 3.75m/s

Wave generator Type: Multi-element (16 paddles) piston type

2. Water channel for two-dimensional models
Osaka Prefecture Univ. (Japan)
L: 12m, W: 0.57m, D: 0.9m.
Wave generator, current generator, forced oscillation mechanism for heave & sway (Heave amplitude: 0.6m, sway amplitude: 1.56m, maximum period: 1.0 sec.)

3. Circulating water channel
Sumitomo Heavy Industries (Japan)
Vertical type with two impellers.
L: 21m, H: abt.8m, B: abt.4.5m.
Measuring section: length: 6.0m, breadth: 2.0m, height:1.75m, water depth:1.40m
Max. Velocity: 3.3 m/sec

4. Ocean Engineering Basin
Institute of Industrial Science, Univ. of Tokyo (Japan)
Built in 2003, Size 50m×10×5m.
Towing carriage speed: 2m/s.
Wind blower speed: 10m/s.
Current generator speed: 0.2m/s, false bottom.
Wave generator: 0.31m × 32 segments plunger type.

5. Seakeeping and manoeuvring basin
MARINTEK (Norway)
Dimensions: L: 40m, B: 6.45m, D: 1.5m
Wave generator: Height: 0.3m, Period: 0.6-1.5sec (irregular waves)
Towing carriage speed: 2 m/s.
5 (6) DOFs forced motions
Current generator speed: 0-0.15m/s.
Typical ship model lengths: 1-3m.

Major Upgrades in Experimental Facilities.

1. Towing Tank (upgraded)
Australian Maritime College (Australia)
L: 100m (previously 60m), W: 3.55m (unchanged), D: 0 to 1.6m

2. Wave generator renewal
National Maritime Research Institute (Japan)
From a flap type to a plunger type in the 400m towing tank
Wave length: 0.5m to 15m, Max. wave height: 0.3m
Generates regular and irregular waves.

3. Guide system for high-speed ships and flying boats
Yokohama National University (Japan)
Tank (existing) dimensions L: 100m, W: 8m, D: 3.5m.
Type: mono-rail under the ceiling.
Maximum Speed = 10m/sec
Directional wave generator (separated plunger type)

4. Installation of a new drive on the carriage
Universities of Glasgow and Strathclyde (U.K.)
The Acre Rd Towing Tank (76m x 4.6m x 2.3m).
Date of commission: December 2004.
Improvement in accuracy and regulation of the set speed (to 0.01%), acceleration and braking.
Maximum speed > 5m/s.

5. Renewal of a 1:14 scale model of the U. S. Navy's Large Cavitation Channel
University of Michigan (U.S.A.)
Date of completion: September 2002
Dimensions: L: 20 ft, H: 10 ft
Test Section .66 ft x .66 ft x 3.0 feet long
Speeds up to 70 knots
Pressurization to 100+ psi

6. A new water filtration system
Webb Institute (U.S.A.)
Installed in a model basin for resistance and manoeuvring tests (28.35 m x 3.04 m x 1.52 m) using copper ions and oxidation to kill algae and sanitise the tank water.

New Model Manufacturing Machines.

1. 5-Axis High speed N.C. milling machine
Hyundai Heavy Industries (Korea)
Date of commission: April 2nd, 2004
Material: Wood/poly-urethane

2. 5-Axis Milling Machine
   Naval Surface Warfare Center (U.S.A.)
   Max. envelope: 10 ft x 20 ft x 2.5 ft
   Materials: Wood, composites, soft metals
   Stereolithography Machine
   Build volume: 20 inch x 20 inch x 23 inch

Major Upgrades in Model Manufacturing Machines.

1. CNC propeller model manufacturing machine
   Memorial Univ. of Newfoundland (Canada)

2. Upgrading from a manual system to a 5-axis CNC system
   Universities of Glasgow and Strathclyde (U.K.)
   Date of commission: December 2004.
   Max. model size: 5m×0.9m×0.9m
   Materials: wood, foam, and MDF (medium density fibreboard).

2.5 Conclusions

There has been steady accumulation of comprehensive experimental data for CFD validation, not only for steady tow cases but also for towing in waves.

Dynamic free surface phenomena such as sprays and wave-breaking still remain difficult to measure. Accurate and reliable skin friction measuring devices are still unavailable.

There has been development of a number of new devices and principles applicable to measurements in marine hydrodynamics, such as magnetic resonance, radar, and GPS.

The role of roughness on frictional resistance has been investigated using a flat plate and a rotating circular cylinder.

A full-scale investigation of drag reduction using micro-bubbles was carried out on a 116m-long ship; a 3% drag reduction was obtained.

A questionnaire was distributed to all the ITTC organizations to inquire new test facilities and new model manufacturing machines since June 2001. It was found that there were 4 new model basins and 1 new circulating water channel, 7 major upgrades in model basins or circulating water channels, 3 new model manufacturing machines, and 2 major upgrades in existing model manufacturing machines.

3. UNCERTAINTY ANALYSIS IN EXPERIMENTAL FLUID DYNAMICS

This section reviews uncertainty assessment in EFD and its utilisation to improve accuracies of both raw data and derived quantities. The raw data are quantities such as mass, length, time, volts and etc. and directly obtained from the instrumentation system. All other quantities resulting from data adjustment or correction to convert the raw data to the appropriate reference conditions are derived quantities. The derived quantities can be used as the starting point in the process of extrapolating full scale results and include non-dimensional quantities such as Reynolds or Froude numbers and dynamic pressure, angles, displacements etc.

3.1 Introduction

The usefulness in quantifying uncertainty for a test is to aid in both designing the experiment and in formalising the expected quality of the outcome so that the customer will know what can be expected from the test. The outcome of any test will be dependent on the entire experimental process. The process includes the design of the experiment, the techniques employed, the instrumentation selected, the flow quality of the facility and both the reduction and presentation of data and data adjustments or corrections to convert the data to the appropriate reference condition (AIAA, 1999). Identification and assessment of significant error sources contributing to uncertainty at each step of the process are thus essential, and will be highlighted in this section, since the definition pertaining to uncertainty and the mathematical tools for estimating uncertainty have been reported in various sources (Coleman and Steel 1999, AIAA 1999, 2003, ASME 1998).

3.2 Error Sources in Measurement Chain

Errors occur at every link in the chain of measurements and the data reduction from the sensor to the final result and are sources of uncertainty. Paying close attention to the major error sources can dramatically simplify uncertainty analyses. In accordance with AIAA Standards (1999), the sources of error can be classified as test techniques, model, flow quality, instrumentation and mathematical model related, as briefly discussed below.

Test Technique. Test technique is closely related to the design of the experiment and must be carefully examined. Some examples of error sources related to test technique are model size and mounting method, dynamometer capacity, turbulence stimulation and model alignment with flow.

Model Shape and Finish. Models are expected to be manufactured to the requisite dimensions, tolerance and surface finish. Improper mechanical fit between a model and dynamometer is an additional error source.

Flow Quality. Corrections required for compensating the effects of flow quality are significant in most cases. Estimation of the effects of flow field distortion and wall interference are important, especially in tests performed at circulating water channels and cavitation tunnels.

Instrumentation. The instrumentation system comprises of the sensors, signal conditioning system and data acquisition system. Calibration can reduce biases but may introduce other errors as does the data acquisition system. The possibility of signal conditioning being a significant influence should not be overlooked either.

Mathematical Model. The most common source of mathematical model uncertainty is the inability of the model to describe exactly the true characteristics of the instrumentation, hysteresis effects and secondary effects of some variables. The mathematical model used to apply correction is an additional source of uncertainty.

3.3 Uncertainty in Measurement

This subsection summarises the recent trend of applying uncertainty assessment in EFD.

PIV/PTV. Quantitative imaging techniques such as particle image velocimetry (PIV), particle tracking velocimetry (PTV) and laser induced fluorescence (LIF) have proven to be effective tools in the study of turbulent flows.

Gui et al. (2001) measured the mean velocity and Reynolds stresses at the nominal wake plane of DTMB Model 5512 in a towing tank with a towed PIV system. The mean velocity is compared with an existing 5-hole Pitot tube data. It is found that PIV uncertainties are about 1% lower than those for 5-hole Pitot tube.

Cowen et al. (2001) reported a single camera coupled PTV-LIF technique applied to measurements in a neutrally buoyant turbulent round jet. Validation measurements show that the uncertainty intervals for the means and rms turbulent intensities are excellent whilst those for the momentum and scalar flux terms were very good.

A method for direct measurement of vorticity (DMV) from digital particle images taken by PIV/PTV is proposed by Ruan et al. (2001). Measurement uncertainty of the DMV method is found to originate directly from the particle image noise and insensitive to the velocity uncertainty.

Lang et al. (1999) constructed out-of-plane velocity components from a set of parallel vector maps obtained by a two-dimensional PIV with application of continuity equation. The smallest uncertainty in cross flow component constructed found by comparing to a CFD results was a RMS error of 3.52%.

A kilohertz frame rate cinemagraphic PIV system, capable of taking up to 8,000 PIV images per second, has been developed by Upatnieks et al. (2002). The two frame cross correlation method employed has a high signal-to-noise ratio and precludes directional ambiguity to allow long uninterrupted image capturing. However, the frame-to-frame alignment uncertainty is a significant source of uncertainty in velocity measurement.

A unique and highly modular and flexible underwater system for stereoscopic PIV measurement has been designed by Pereira et al. (2003). The system is intended for planar three-dimensional velocity measurement in large facilities such as towing tank or tunnels and errors under 2% for in-plane components and 4% for out-of-plane ones are reported.

Bias and precision errors of digital particle image velocimetry (DPIV) were quantified by Forliti et al. (2000). Bias errors are found to exist for all sub-pixel peak finding algorithms and the presence of the biases tend to affect the precision error. Calibration was effective in removing the bias errors in the potential core of a rectangular free jet but less effective in the shear layer. The study also stressed the need for in situ quantification of DPIV uncertainty.

Zhang et al. (2002) estimated the uncertainties of stereoscopic particle image velocimetry (SPIV) measurement and found that the angular-displacement method can provide up to 40% higher out-of-plane accuracy than the translation method.

The accuracy of a 3D PTV algorithm was tested by Kieft et al. (2002) to examine its capability to locate particles in 3D space. It appeared that the algorithm can track the particle position with an accuracy of less than 0.5 camera pixels. The calculated velocity vectors for particles located in a 2D plane have a standard deviation of less than 1%.

Christensen et al. (2002) developed a particle image accelerometry (PIA) to measure Eulerian acceleration field by time-differencing successive measurements of PIV. It is found that the fluctuating error in velocity components is the order of 0.15 pixels and that the random error is proportional to the particle image diameter for constant image density. A methodology is presented that allows these errors to be removed from the acceleration statistics based upon the zero-time-delay velocity measurements.

McKenna et al. (2002) investigated several traditionally based techniques for cross correlation image processing in terms of computational efficiency and measurement accuracy. It was found that direct spatial domain correlation is more accurate than FFT based methods. The use of the more robust methods (e. g. dynamic FFT) can improve the accuracy of the FFT significantly.

<u>Pitot Probe.</u> Olivieri et al. (2003) investigated flow structures under bow breaking wave generated by a model of fast displacement ship. Wave heights were measured by a finger probe and mean velocity fields with a 5-hole Pitot tube. Uncertainties of the measured velocity components u, v and w were found to be 0.69%, 1.47% and 1.58%, respectively.

<u>LDV.</u> Martin et al. (2000) measured the wake of a scaled helicopter rotor operating in hover. It was suggested that a quantitative estimate of source of LDV measurement bias, especially gradient bias, should be included and the ratio of effective length of the probe volume to the smallest expected vortex core radius should remain less than 5% to minimise gradient bias.

DeGraaff et al. (2001) developed a high resolution laser Doppler anemometer (LDA) with an experimental method for measuring and removing velocity bias and for estimating the absolute accuracy of the LDA measurement. Uncertainty estimates for the mean velocity and Reynolds stress measurements of a flat plate 2D turbulent shear flow were given.

Buttner et al. (2003) presented the advantages in employment of multimode fibres (MMF) for beam delivery in LDA. He demonstrated that employment of multimode-light can reduce the length of the measurement volume to a few percent of the length of two laser beams. The variation in the fringe spacing also drops under 0.05%.

Hot wire. Lavoie et al. (2003) implemented four data-reduction schemes to measurements taken in the near field of a free, round jet with single-, cross- and four-wire hot wire probes. Uncertainty analysis of the four wire probes indicated that they can simplify the study of three-dimensional flows by eliminating correction equations for off-axis velocities, alignment requirements and multiple sampling orientations but the probe's acceptance domain is not large enough to allow for accurate measurements in high turbulence intensity flows.

Petrovic et al. (2003) analysed the measurement accuracy of a different hot-wire probes processing between two and 12 sensors. It was found that neglecting the instantaneous fluctuations of the velocity gradients for the measurement of the cross-stream velocity component has a crucial influence and results in large errors.

Pressure Sensitive Paint. Low-speed (M<0.1) Pressure Sensitive Paint (PSP) measurements can be quite difficult owing to the high SNR required but reasonably accurate PSP measurements can be made even at very low flow speeds when care is taken (Bell et al. 2001). Brown (2000) reported the measurement results for an NACA 0012 airfoil at an angle of attack of 5 degrees and a free stream velocity of 20m/sec. The error in surface pressure was 16% probably because the airfoil did not fully span the test section width and formation of the wing-tip vortices.

Balance. Reis et al. (2002) described a method for the estimation of uncertainty in measurements of aerodynamic loads using an external balance and reported a recent assessment of the friction forces and corresponding uncertainties in order to better evaluate the contribution. (Reis et al. 2003)

Methodology. Meyn (2000) described an alternative method wherein the uncertainty in each measurement is represented as a vector of elemental uncertainties and these vectors are used to calculate uncertainty in the final results. The method is mathematically equivalent to the traditional method but eliminates the need for a covariant matrix, is easier to automate, and makes complex uncertainty correlations easier to understand.

3.4 Uncertainty in EFD Data used for CFD Validation

The purpose of the CFD Workshop Tokyo 2005 (CFDWS 2005) was to enhance research and development and to assess the state of the art of numerical ship hydrodynamics. The experimental data used in the workshop, however, will also serve as good examples of uncertainty analyses on derived quantities.

The hull forms subjected to test computations were the KRISO Container Ship (KCS), the Modified KRISO VLCC2 (KVLCC2 M) and a US Navy Combatant (DTMB5415), identical to those used in the Gothenburg 2000 Workshop. Test computations of the CFDWS 2005 are not confined to resistance and self-propulsion cases as before but expanded to account for oblique motion, diffraction, and free sinkage and trim conditions as shown in Table 3.1. New experiments were performed to obtain an additional database for validation through rigorous uncertainty analyses. The results along with those previously used are summarised in Table 3.2.

Table 3.1- Test conditions of CFDWS 2005.

Test no.	Ship name	Test Condition
1.1	KCS	Fixed
1.2	DTMB5415	Fixed
1.3	DTMB5415	Free (F)
1.4	KVLCC2M	Double Body (DB)
2	KCS	Self propelled (SP)
3	KVLCC2M	Obliquely towed (OT)
4	DTMB5415	Diffraction
5	KVLCC2M	DB (Grid dependency)

Table 3.2- Uncertainty in Derived Quantities.

Derived quantity	Ship name	Uncertainty
Total Resistance Coefficient (C_T)	KCS	1.0
	DTMB5415	2.2
	KVLCC2M	0.7
Wave Profile (ζ)	KCS	0.92
	DTMB5415	6.4
Longitudinal Wave Cut (ζ)	KCS	0.43
	DTMB5415	4.7
Velocity downstream of Propeller Plane (u, v, w)	KCS	(0.38, -, -)
	KCS (SP)	(5.0, -, -)
	DTMB5415	(5.3,7.2,16.8)
	KVLCC2M	(0.5, 1.0, 1.0)
Sinkage & Trim	DTMB5415 (F)	(1.4, 1.83)
Side forces(x, y)	KVLCC2M (OT)	(14.1, 8.66)

It is worthy to note that Kume et al. (2005) measured and analysed the uncertainties of hydrodynamic forces, wake, surface pressure distribution and side forces of an obliquely towed KVLCC2M model. The uncertainties are not negligible in many cases and decomposition and quantification of error indicate that the standard error of estimate of a calibration line of a current meter is a dominant source of error in wake measurements.

3.5 Conclusions

There is a growing trend of applying UA in EFD and now an increasing number of EFD papers are dealing with UA, especially where relatively new measurement techniques such as PIV/PTV or LDV are involved.

The implementation of UA in ITTC community is also growing after the implementation of ITTC standard procedures. In the Gothenburg 2000 CFD workshop, UA was applied to the measurement of wave resistance, wave height and wake in the propeller plane of the model ships such as KCS, KVLCC2 and DTMB 5415 for the purpose of CFD validation and verification. In CFDWS Tokyo 2005, the application of UA was expanded to include oblique towing, diffraction and free-sinkage and trim towing conditions.

The tendency, however, is not yet widespread, and is still confined to specific fields. Further efforts to encourage UA in EFD in general are necessary since the identification of error sources and the quantification of their contribution to the final result are important for designing the experiment and formalising the expected quality of the measurement.

4. TRENDS IN COMPUTATIONAL FLUID DYNAMICS

4.1 Introduction

This chapter summarises the ongoing research effort toward the development of efficient numerical tools in the area of computational hydrodynamic analysis and design of ships, reporting trends in research and experience in industrial applications as emerged from the literature of the last three years. The section opens with a summary of the main results and conclusions of the CFD Workshop Tokyo 2005 (CFDWS05), Section 4.2. This is followed by Sections on practical applications of CFD, Section 4.3, progress in computational methods that have evolved over the last three years, Section 4.4, and new application areas that are being pursued, Section 4.5.

4.2 Results of CFDWS Tokyo 2005

Organised by Hino (chair), Hinatsu, Hirata, Kodama, Stern and Ukon and held in Tokyo, Japan, in March 2005, it is the fifth workshop of the series (Gothenburg 1980, 1990 and 2000, Tokyo 1994). In the following a brief summary of the objectives of the Workshop, the participating groups, results on V&V, predicted quantities such as resistance, wake-fields and viscous flow, free surface pattern and most relevant issues raised during the Workshop will be given.

Objectives of CFDWS05. CFDWS05 was prepared with the aims of assessing the state of the art in computations of viscous flows around a ship hull and accelerating the research and development of numerical ship hydrodynamics. The set of test cases for the benchmark ships has hence been updated from the Gothenburg 2000 Workshop (G2K) to include free sinkage and trim, oblique tow and wave diffraction effects in head seas. In addition, an effort has been made to evaluate the use of the ITTC Verification and Validation (V&V) procedure for uncertainty analysis in CFD (ITTC Quality Manual 7.5-03, 2002).

The Workshop was organised into 5 test cases: 1) container, combatant and tanker hull forms undergoing steady tow conditions in still water or without free surface; 2) container hull in self propulsion conditions in still water without free surface; 3) tanker hull in oblique tow conditions without free surface; 4) combatant hull in tow conditions in head waves and 5) tanker hull in steady tow with a defined grid resolution study. Information about the test cases, benchmark geometries and flow conditions may be obtained at the URL http://www.nmri.go.jp/cfd/cfdws05/.

Benchmark Hull Forms. Three hull forms were selected for the computations. The VLCC tanker designed by the Korean towing tank KRISO (version 2, named KVLCC2) with modification (named KVLCC2M). New measured data for this hull at zero incidence and for oblique tow were provided by Kume et al. (2005). The KRISO container ship (named KCS) for which measured data (with and without propeller) are available from Van et al. (1997, 1998), Lee et al. (1998), Fujisawa et al. (2000), Tsukada et al. (2000) and Kume et al. (2000). The David Taylor Model Basin (DTMB) design of a navy combatant Model 5415 (named DTMB 5415) for which measured data is available from Ratcliffe (2000), Olivieri et al. (2001), Gui et al. (2001 a,b,c).

Participants and Methods. A total of 20 groups participated in the Workshop, from 26 organisations and 13 countries using 17 different flow codes. The details of each code were collated using a questionnaire to assist in the classification of the numerical methods.

All of the methods (except one) computed the flow around the hull using Cartesian coordinates and solved the Reynolds Averaged Navier-Stokes (RANS) equations for the velocity and pressure with various turbulence models. The exception used a vorticity transport panel method. The turbulence models ranged from zero equation to Reynolds stress models with the majority using two equation SST k-ω or variants. Near wall and wall function models were used. Surface tracking and surface capture methods were used for the free surface with the majority using surface capture methods. Body force methods and the actual propeller geometry were used to define propeller characteristics. The majority of methods were based on finite volume schemes. At least second order methods were used throughout for the velocity convection terms.

Grid generation methods varied considerably with block structured, multi-block structured and block overlapping methods predominating. Two participants used unstructured methods. The number of grid points varied from around 1 million to 10 million with the majority in the range 2-4 million points.

Verification Results. The variation coefficient V ($V=100\sigma/C$ where σ is the standard deviation of all the results obtained for each test case conditions and C is the relevant coefficient) for the resistance components for the three geometries was found to be reasonably small for C_T and C_F but for C_p it was higher. The values obtained were 4.2%, 11.5% and 6.6% for C_T, C_P and C_F respectively for the KCS, 7.1%, 27.6% and 5.9% for the DTMB 5415 and 6.9%, 24.9% and 6.4% for the KVLCC2M. The values for C_p have been shown to be particularly grid-dependent. However, when compared with the oblique tow test case, the equivalent C_x variation is around 10% for the KVLCC2M. This may be explained by the use of reduced number of average grid points due to the lack of symmetry. The variation of the results for the propulsion characteristics illustrates good agreement between the methods. The largest variation was for the prediction of K_Q of 12%. The variation of the results for the wave diffraction case showed reasonable agreement for the 0th and 1st amplitude of C_T and for the 0th amplitude for C_H. However considerable variation was obtained for the phase of all the coefficients. The numerical uncertainty of the results for the KVLCC2M was examined in detail in test case 5 with uncertainties ranging from 90% for the coarsest grid to 0.6% for the fine grid.

Validation Results. Validation of the results for the total resistance, C_T was obtained for each of the three geometries for test case 1, for the propulsion characteristics for test case 2, for the manoeuvring coefficients C_X, C_Y and C_N for the oblique tow test case 3 and for the amplitudes and phases of the forces and moments C_T, C_H and C_M for test case 4 together with velocity, pressure and wave profiles where appropriate. The average comparison error E (the difference between the computed value and the measured value) for test case 1 is 3.1% for KCS, 5.1% for DTMB 5415 and 6.1% for KVLCC2M.

Predicted Wake Fields and Viscous Flow. Computed velocity field contours compare quite well with the measured data. The hook-shape for the KVLCC2M is particularly well resolved by the results obtained from Reynolds stress based turbulence models and by models that are calibrated for such flows. Results for the oblique tow case compare remarkably well with the position and strength of vortices being well captured. Detailed examination of the velocity components shows some discrepancies that require further evaluation.

Predicted Free Surface. Comparison with the wave profiles on the hull surface show good agreement with measured data. All of the methods showed reasonable capture of wave crests near the hull. The data from longitudinal wave cuts show somewhat greater dependency on grid resolution and numerical scheme.

Propeller Hull Interaction. The overall trends of the propulsor wakes were reasonably well captured by the different methods. The change in the pressure field caused by the propeller appeared to correlate well with the measured data.

Wave Diffraction. The overall trends of the wakes obtained from the wave diffraction case agreed well with measured data. The zero-th and first amplitudes and phase of the wave elevations for the wave diffraction case showed good agreement with the measured data.

Comparison with G2K. Comparing the results obtained at CFDWS05 with those from G2K there has been a slight improvement in the variation of prediction of resistance coefficients (from 5%-8% for G2K to 4%- 7% for CFDWS05), however the average comparison error E appears around the same value. Predictions of the viscous wake indicate similar trends with the best results obtained from Reynolds stress models or those models that are calibrated for such flows. Generally the predictions of the free surface wave patterns have improved, especially in terms of the resolution of the wave crests downstream of the stern.

4.3 Practical Applications of CFD

Inviscid Flow Calculations. Despite all the advances with RANS for surface ships one should not neglect the use of simpler models where appropriate. For example, simple regression analysis formulas have been created by Sahoo et al. (2004) for slender catamaran hull forms, which compare well with model scale experimental data. Andrewartha et al. (2003) extended an inviscid based method to predict resistance of foil assisted catamarans. Doctors and Scrace (2003) show that classical linearised theory can be used to predict steady state trimaran resistance and how it can be varied with the position of the side hulls.

Mixed Viscous/Inviscid Calculations. There is still much use of mixed/hybrid methods where the RANS code is used in a small region. Such a mixed method is used by Chen and Huang (2002) for the inverse design of the propeller inflow for a container ship. Using SHIPFLOW the RANS portion of the calculation was confined to the stern half of the ship with potential flow and boundary layer methods used elsewhere. In a similar vein Raven and Starke (2002) use an inviscid panel method for computing the free-surface. A grid is then generated to conform to the body and the inviscidly computed free-surface and RANS computations performed for this domain.

A truly coupled method has been presented by Guillerm and Alessandrini (2003) where near the ship RANS equations are solved for the viscous flow field while the outer domain is computed with the Fourier-Kochin formulation of the Green function to compute the potential flow. The two regions are strongly coupled through the velocity, pressure and free-surface elevation. For similar reasons Hänel et al. (2003) coupled a three-dimensional finite volume solver, using level sets for the free surface, in the vicinity of a ship with a shallow water Boussinesq method in reduced dimensions in the far field.

Viscous Flow Computations at Model Scale. Progress continues to be made in the computation of ship viscous flows using RANS for the complete configuration. Many computations have been done of the hull forms chosen for the Gothenburg 2000 (G2K) workshop (e.g. Larsson et al., 2003) and the CFDWS Tokyo 2005. As progress has been made in application to ship hulls they have also become increasingly more sophisticated with the inclusion of more features of the real geometry. This is demonstrated by Burg et al. (2002) and Gorski et al. (2004) for Model 5415 with shaft and struts. Bull et al. (2002) computed the flow over the fully appended NATO research vessel "Alliance" including shafts, struts, and rudder. Hino et al. (2003) also computed the flow past a twin propeller coastal ferry with shafts, struts, and rudder.

Tracking methods can be used very effectively for free-surface predictions. Gorski et al. (2002) used a surface tracking approach for a tumblehome geometry with wave piercing bow and demonstrated very good comparison of measured wave height on the hull as well as in the stern region directly behind the model. There is room for improvement with tracking methods as demonstrated by Raven et al. (2004). By removing any time dependence from the equations and strictly solving a steady form of the problem robust free surface RANS computations are achievable with only a few iterations on the computed free surface height. Comparisons of wave heights with Series 60 and the Dyne tanker are in good agreement with experimental data.

There has been a move toward the use of capturing methods for dealing with the free-surface prediction by many groups. The capturing methods have continued to become more accurate and can provide good direct estimates of the wave field and total resistance as demonstrated by Park et al. (2004) using a level set approach for the KRISO KCS container ship. Di Masco et al. (2003) performed a number of calculations using tracking and a single phase level set technique and demonstrated wave height predictions with the level set are just as accurate as tracking predictions.

Many RANS ship computations have been with prescribed sinkage and trim. One effort that demonstrated very good predictions of resistance over a speed range for a frigate as well as two ACC hulls is that of Jacquin et al. (2004). The RANS computations were done with ICARE, which imposes fully nonlinear free surface conditions with a tracking approach. The computations provide very good correlation with experimentally measured values in a towing tank. Using a VOF method with COMET Azcueta (2004) demonstrates the ability to use RANS to accurately compute resistance for a monohull littoral combat ship in a Froude number range of approximately 0.25 to 0.9. Azcueta (2003) also demonstrated similar results for a high speed planning craft for a range of Froude numbers from 0.75 to 4.0.

Viscous Flow Computations at Full Scale. There is continued interest in predicting full scale effects with RANS codes. Although there is little high quality experimental data a recent initiative known as EFFORT (European Full-scale Flow Research and Technology) started whose principal objectives are the refinement and validation of CFD prediction methods for the viscous flow around a ship at full scale, Verkuyl and Raven (2003). As discussed by Bull et al. (2002) full scale measurements were conducted on two hull forms: the Dutch frigate the "DeRuyter" and the NATO research vessel the "Alliance". Computed results are shown for the "Alliance" both at model and full scale. Tahara et al. (2002) demonstrate for the Series 60 that CFD predictions of the boundary layer shrinkage from model to full scale is not consistent with simple scaling laws. In addition, they demonstrate pressure differences due to scale also. The sensitivity of details of the flow to Reynolds number is also demonstrated by Choi et al. (2003) for a 300,000 TDW VLCC and Duvigneau et al. (2003) for the KVLCC2 tanker where it has previously been shown that details of the flow at the propeller plane include a strong vortical structure at model scale. However, at full scale the wake is predicted to be much less severe due to the weaker bilge vortices. For configurations where the propeller is mounted on a shaft/strut system the differences between model and full scale can also be significant. As demonstrated by Gorski et al. (2004) the dominant feature entering the propeller is the wake of the shaft/strut directly in front of it. At model scale this wake is impacted by the hull wake, but at full scale it is not.

An attempt at including surface roughness is the combined experimental/numerical approach of Leer-Andersen and Larsson (2003) where the original skin friction coefficient in the boundary layer method of ShipFlow is modified to account for the increase in skin friction for full scale ships due to surface roughness or bio-fouling. This is accomplished through the inclusion of an efficiency parameter of the surface, which must be measured.

4.4 Progress in Viscous Methods

Grid Types. Accuracy of solutions depends particularly on the quality of the grid used for the calculation. Structured grids have been the workhorse for many of the marine vehicle calculations to date. However, it can be very difficult to generate high quality grids for complex structures such as shaft and strut arrangements as discussed by Gorski et al. (2004) and for dealing with other complexities such as the inlets of waterjets (Ebert et al., 2003). To deal with such geometries there is a clear trend toward using Chimera or overset grids, or unstructured grids in many instances.

Significant progress continues to be made in chimera or overset gridding techniques and the associated software that are needed with it (e.g. Chan et al., 2002). Simonsen and Stern (2003) used the approach for the Esso Osaka where an overlapping grid is generated around the rudder and joined to the hull grid with a chimera approach. For better resolution of the

Kelvin wake field of a surface combatant Wilson et al. (2004) used an overset approach to embed a fine grid specifically for better free surface resolution.

Unstructured grids offer a tremendous amount of capability for handling complex geometry such as shafts and struts as demonstrated by Burg et al. (2002) and Hino et al. (2003). Traditional tetrahedral grids, which offer the most flexibility, have often had difficulty providing good resolution of boundary layers and are not considered as accurate as structured grids for them (Luo et al., 2000). This led to much development of hybrid grids, which often consist of prisms near walls for boundary layer resolution with tetrahedral grids away from the walls. With these hybrid grids it is not clear that there is a loss of accuracy from traditional structured grids. A comparison of structured and unstructured grids for an axi-symmetric body by Dailey and Ebert (2003) showed negligible differences when compared with measured skin friction data.

Unstructured grids lend themselves readily to grid adaptation. Lohner et al. (2004) have shown how adaptive unstructured meshes can better represent complex flow details such as the air water interface. This is also demonstrated by Hay et al. (2004) for a Wigley hull computation where a local refinement and coarsening strategy is used in an unstructured RANS solver to better capture the free surface. One issue that comes up with modern parallel processing and adaptive gridding is the load balancing between processors as the grid changes during the computation. Progress is also being made in this area as demonstrated by Cavallo et al. (2004).

An area getting renewed interest is the use of traditional Cartesian grids. With these methods a Cartesian grid of the entire domain is generated and then the body surface is used to cut cells that intersect it to preserve the geometry features. The grid generation algorithm can be completely automated. Wang and Chen (2002) demonstrate the method for aircraft and missile configurations. Dommermuth et al. (2004) demonstrate Cartesian grids with interface capturing methods and the use of adaptive mesh refinement near the interface.

Free Surface Treatment. Currently, a variety of methods are being used for computing the free-surface interface for surface ship calculations. At G2K approaches were about equally split between tracking methods and capturing methods. The traditional tracking methods have been problematic for steep or breaking waves.

One issue with capturing methods, which are of the volume of fluid or level set variety, is the smearing of the free surface depending on grid resolution. This can be addressed somewhat by better resolution of the free surface interface. However, it is also important to include good numerics to best predict the interface with a minimum number of cells. Hay et al. (2004) have compared several discretisation schemes and their influence on interface smearing. Their results indicate compressive discretisation schemes with an adequate number of points near the interface can provide high quality results for the interface location.

Capturing methods tend to more naturally deal with transom sterns changing from wet to dry conditions as shown by Di Masco et al. (2003) for Model 5415. Various researchers have also shown that capturing methods can inherently predict breaking wave phenomena. Hino (2004), using a single phase level set approach, can predict wave breaking for a ship model with a blunt bow. Broglia et al. (2003) also used a steady single phase level set approach to investigate wave breaking around an infinitely long wedge. The calculations are shown to do well predicting the contact lines, plunging jet shape, angle and velocity impact as compared to experimental data. Sethian and Smereka (2003) recently provided an overview of level set methods.

An alternative to level set and VOF methods is the Constrained Interpolation Profiles

(CIP) scheme developed by Yabe et al. (2001). This method uses a compact upwind scheme for the convection terms and a pressure-based algorithm that can treat liquid, gas and solid phases by using a single set of equations. Hong et al. (2003) used a modified level set approach with CIP for computing the flow about a ship in a restricted waterway. Comparisons with experiments showed that the approach worked well for predicting wave height, wave breaking, and the time when wave breaking starts.

Turbulence Modelling. Most calculations for marine problems are done with conventional existing one- and two-equation models. Perhaps, the most progress has occurred in the application of full Reynolds stress models. There is evidence from G2K that full Reynolds stress models perform better at predicting the strong vortical flow at the propeller plane of the KVLCC2 tanker. This is demonstrated by Duvigneau et al (2003) who show that at model scale it is necessary to use 2nd order closures for the accurate prediction of this vortical flow.

Methods for the Numerical Solution. Imas et al. (2003) extended the Smoothed Particle Hydrodynamics (SPH) approach to three-dimensions and the application of a surface piercing double wedge with bow flare and rake angles. The approach can capture salient features of the flow, but details are under resolved. Although it is possible to apply SPH to realistic three-dimensional flows according to Imas (2004) a two order of magnitude increase in resolution is needed as compared to comparable VOF methods to capture flow details.

LES Methods. The Large Eddy Simulation (LES) of various flows continues to become more practical and less of a research area. This is emphasised by the work of Alin et al. (2003) and later Bensow et al. (2004) who demonstrate LES for a fully appended submarine model at a Reynolds number of 12 million. Lillberg and Svennburg (2004) also computed Model 5415 with LES at a similar Reynolds number and Chun et al (2004) have performed LES computations for the Wigley and Series 60 hull forms at Reynolds number of approximately 4 million.

4.5 New Applications

Propulsor/Hull Interaction. To account for propeller methods vary from simple actuator disk models to computations with rotating propellers. Takada et al. (2002) demonstrate accurate predictions of resistance and self-propulsion around the KCS container ship using lifting surface theory for the propeller. Bull et al. (2002) use a lifting body panel method for propeller forces that are distributed on the RANS grid cells for their fully appended computation of the "Alliance". Hino et al. (2003) used an unstructured grid to include tractor and pusher pods on a coastal cargo ship with an actuator disk model for the propeller effect. Hino et al. (2003) also compute the flow past a twin propeller coastal ferry with shafts, struts, and rudder and perform self propulsion calculations.

Computations which include the operating propeller as part of the full computation have been demonstrated by Lübke and Mach (2004) for the KRISO KCS container ship. For these calculations the propeller geometry is included in full detail as part of the full hull computation. Burg et al. (2002) included a fully operating propeller for Model 5415 in addition to the shafts, mounting struts and rudder.

An interesting example of an integrated propulsor/hull is the ducted configuration discussed by Gorski et al. (2002). This tumblehome geometry has a wave piercing bow with a sonar dome that generates two vortices that run the length of the hull. These bow generated vortices run directly into the ducted propulsors, which are mounted flush to the hull.

Waterjets. RANS computations have been shown to be quite accurate at predicting the inflow for two waterjet configurations by Ebert et al. (2003), which included the Athena hull configured with a traditional waterjet and a tumblehome geometry fitted with an advanced waterjet with below water exhausts. Bulten and Verbeek (2003) also used three-dimensional CFD calculations to aid in the design of the inlets for a fast ferry, a patrol boat, and a high speed motor yacht.

4.6 Conclusions

Computational capabilities are making inroads in the design processes for many vehicles of interest including marine vehicles. A brief overview of how computer simulations have changed for marine engineers, and are likely to change in the future, is given by Bucknall and Greig (2002). Inviscid methods are still often used, but RANS codes are starting to play a larger role in the study of viscous flow fields generated by marine vehicles. It seems inevitable that RANS will have an even larger role in the future as computer power increases and the application of such codes matures even further. However, it will still take considerable effort to have the confidence in the RANS codes that currently exists with the model tests.

5. UNCERTAINTY ANALYSIS FOR TOWING TANK MEASUREMENTS

5.1 Introduction

When the value of a quantity is determined experimentally, as it is done in towing tanks, one of the main goals is to know the closeness of the agreement between the determined and the true values. Error is the difference between the experimentally determined value and the truth. Errors can be considered to be composed of two components: a precision component, which contributes to the scatter of data, and a bias component, including the systematic and non random errors. Due to the assumption of these experimental errors, the final result of a test has to be corrected with the error interval.

In order to determine uncertainties, the 24th ITTC Resistance Committee decided to propose a worldwide series of comparative tests for identifying facility biases under the framework of ITTC procedures for uncertainty analysis and invited all the ITTC Members to participate.

It was not the first time that this kind of worldwide series were proposed. In the 9th ITTC in 1960, some institutions agreed to test standard models, possessed by the institutions, in the same conditions to determine “erratic variation of model resistance” and study “techniques of measuring model resistance”. In those tests error sources in the measurements were detected but the errors were not determined, data was lost and model’s dimensions were not controlled. In the 17th ITTC in 1984 a cooperative experimental program to produce a comprehensive database was proposed. In this case total resistance, wave pattern resistance and spectra, wake survey, hull pressure, hull shear stress and boundary layer transverses were measured using 4 models Wigley hull, Series 60, Athena and a tanker. Ten institutions did these tests and obtained huge amounts of data but the tests did not allow the determination of facility biases because the institutions did not test the same models and the tests were not systematised. For this reasons, a new worldwide series of comparative tests was proposed to collect the necessary data to do the uncertainty analysis in experimental fluid dynamics of the participant institutions.

The Resistance Committee has created the following technical procedure for identifying facility biases, compiling models and test procedures information. For these tests two geosims DTMB 5415 Combatant, with 5.720 and 3.048 meters length respectively are used.

5.2 Model Definition

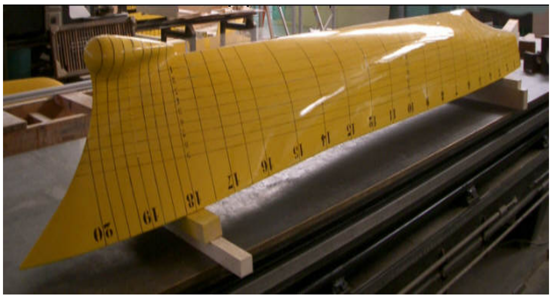
In order to minimise the influence of the model geometry in facility biases identification the institutions participating in the worldwide series for identifying facility biases have to test the same model that will travel from one institution to another.

Due to the different model scales used in the towing tanks and in order to increase the number of institutions participating in the ITTC worldwide series for identifying facility biases, two geosims of the model DTMB 5415, with 5.720 and 3.048 meters length respectively, will be used. Two groups of institutions have therefore been formed for testing and analysis.

Geosim A: 5.720 Metres Length. CEHIPAR model 2716 is a wooden geosim of DTMB 5415 model. The scale of the model compared with the real vessels is 24.824. Model length between perpendiculars (L) is 5.720m and the calm water draft (T) to be used in the tests is 0.248m. The model will be tested without initial trim. The model displacement in the calm water draft of 0.248 m without trim is 549 Kg, in fresh water.

The above photographs show the model.

Geosim B: 3.048 Metres Length. Geosim B has been made of wood at CEHINAV. The scale of the model compared with the real vessels is 46.6. Model length between perpendiculars (L) is 3.048 m. and the calm water draft (T) to be used in the tests is 0.132 m. The model will be tested without initial trim. The model displacement in the calm water draft of 0.132 m without trim is 83 Kg, in fresh water.



5.3 Test Definition

Facility biases will be analysed for the following most typical towing tank tests:

- Resistance
- Sinkage and trim
- Wave profile and wave elevations

It is compulsory to measure resistance but it is not compulsory to measure all the data. Each institution can decide, according their capabilities, to measure sinkage, trim, wave profile and wave elevations. Nevertheless all the institutions are encouraged to measure all data as possible in order to have enough data to determine facility biases.

Each institution will test the model on 4 different days in order to change the test conditions and obtain better uncertainty analysis results. They will use their standard techniques to test the model and must correct their results taking into account the blockage effects using their standard procedures. Notice that if due to their standard procedures to correct blockage effects the speeds have to be changed, the institutions will have to do that and indicate the velocity variation in the submitted data. In each session, 3 different velocities, corresponding Froude numbers 0.10, 0.28 and 0.41, will be tested using the following schedule:

Table 5.1- Froude numbers of the runs.

Fr	Day 1	Day 2	Day 3	Day 4
Speed 1	0.28	0.28	0.28	0.28
Speed 2	0.10	0.10	0.10	0.10
Speed 3	0.28	0.28	0.28	0.28
Speed 4	0.41	0.41	0.41	0.41
Speed 5	0.10	0.10	0.10	0.10
Speed 6	0.28	0.28	0.28	0.28
Speed 7	0.41	0.41	0.41	0.41
Speed 8	0.10	0.10	0.10	0.10
Speed 9	0.28	0.28	0.28	0.28
Speed 10	0.41	0.41	0.41	0.41

Table 5.2- Testing velocities in m/s. Geosim A.

V (ms^{-1})	Day 1	Day 2	Day 3	Day 4
Speed 1	2.096	2.096	2.096	2.096
Speed 2	0.749	0.749	0.749	0.749
Speed 3	2.096	2.096	2.096	2.096
Speed 4	3.070	3.070	3.070	3.070
Speed 5	0.749	0.749	0.749	0.749
Speed 6	2.096	2.096	2.096	2.096
Speed 7	3.070	3.070	3.070	3.070
Speed 8	0.749	0.749	0.749	0.749
Speed 9	2.096	2.096	2.096	2.096
Speed 10	3.070	3.070	3.070	3.070

Table 5.3- Testing velocities in m/s. Geosim B.

V (ms^{-1})	Day 1	Day 2	Day 3	Day 4
Speed 1	1.530	1.530	1.530	1.530
Speed 2	0.547	0.547	0.547	0.547
Speed 3	1.530	1.530	1.530	1.530
Speed 4	2.241	2.241	2.241	2.241
Speed 5	0.547	0.547	0.547	0.547
Speed 6	1.530	1.530	1.530	1.530
Speed 7	2.241	2.241	2.241	2.241
Speed 8	0.547	0.547	0.547	0.547
Speed 9	1.530	1.530	1.530	1.530
Speed 10	2.241	2.241	2.241	2.241

Using the first schedule and taking into account only the last 9 runs of each session (not shadowed in the table) each institution will obtain 12 resistance values for each velocity. Additionally, if the water is especially calm at the beginning of each session, the four resistance values corresponding to the first run of each session will allow the comparison of the effect of the water movement in the results.

Each institution will do the tests following their usual procedures with the best possible care in order to obtain good quality results. Thus all measurement equipments and systems have to be properly calibrated and prepared.

Resistance. For the resistance tests the model has to be free to heave, pitch and roll but must be restrained in surge, sway and yaw.

Towing tank water temperature has to be measured daily at the model mid draft, from which density ρ and kinematic viscosity ν are linearly interpolated using fresh water values as recommended by ITTC Quality Manual 7.5-02-01-03 "Testing and Extrapolation Methods, General, Density and Viscosity of Water".

The measurement system has to be described, specifying all the measurement systems, conditioners, including the calibration curves and the frequency of the data acquisition.

The comparison variable is defined for residuary C_R resistance as given in the following data-reduction equations:

$$C_R = C_T^{Tm} - C_F^{Tm}(1+k) \tag{5.1}$$

where,

$$C_F = \frac{0.075}{(\log_{10} Re - 2.0)^2} \tag{5.2}$$

is obtained by the ITTC 57 formula, where k is the form factor and

$$C_T^{Tm} = \frac{M_x^{Tm} g}{0.5 \rho U_c^2 S} \tag{5.3}$$

where,
T_m indicates the temperature of measurement.

C_R is selected as the comparison variable for resistance. Furthermore, $C_T^{15\deg}$, defined as

$$C_T^{15\deg} = C_R + C_F^{15\deg}(1+k) \qquad (5.4)$$

is also used as a reference quantity for the uncertainty assessment, since it calibrates all data to the same temperature T=15°C.

The form factor *k* has to be calculated using Prohaska's method, as recommended by ITTC Quality Manual Procedure 7.5-02-03-01.4 "Performance, Propulsion, 1978 ITTC Performance Prediction Method".

Sinkage and Trim. The running sinkage Z_{VM} is obtained as the mean value of the running sinkage in the fore perpendicular Z_{VF} measured in hull section 20 and the running sinkage in the aft perpendicular Z_{VA} measured in section 0.

$$Z_{VM} = \frac{Z_{VF} + Z_{VA}}{2} \qquad (5.5)$$

The comparison Z_{VM} value has to be expressed in metres.

The running trim angle θ_D has to be obtained using the following formula:

$$\theta_D = \tan^{-1}\left(\frac{Z_{VF} - Z_{VA}}{L}\right) \qquad (5.6)$$

The comparison θ_D value has to be expressed in radians and is positive bow up.

Wave Profile and Wave Elevations. Wave profile on the hull surface and a vertical longitudinal wave cut in a plane separated 0.172·L from the centre plane have to be obtained for all the testing cases.

Data for each wave profile will contain pairs of coordinates (x, z) where x is the longitudinal distance of the measurement point referred to the aft perpendicular (section 0) and z is the wave elevation over the calm water plane. Coordinate x is positive from stern to bow and coordinate z is positive from the calm water plane up.

The coordinates corresponding to the wave on the hull surface will be named (x_h, z_h) and the corresponding to the wave cut will be named (x_c, z_c).

The wave cut must cover at least the distance from the fore perpendicular to the aft perpendicular. The comparison coordinates x_c, z_c, x_h and z_h must be expressed in metres.

5.4 Uncertainty Analysis Procedure

The goal of the comparison is to obtain for each test type, each velocity and each institution the percentages of the precision limit (P), bias limit (B) and uncertainty (U) as defined in 7.5-02-01-01 "Testing and Extrapolation Methods, General Uncertainty Analysis in EFD, Uncertainty Assessment Methodology".

The procedure to analyse the data is explained in "IIHR technical report T442, Statistical Approach for Estimating Intervals of Certification or Biases of Facilities or Measurement Systems Including Uncertainties" by F. Stern, A. Olivieri, J. Shao, J. Longo, and T. Ratcliffe.

Resistance, sinkage and trim uncertainty analysis will be made using the following variables:

$C_T^{15\deg}$ Resistance comparison variable.
Z_{VM} Sinkage comparison variable.
θ_D Trim comparison variable.

Wave profile on the hull surface, (x_h, z_h), and the wave cut (x_c, z_c) recordings will be analysed in their main maximum and minimum values. So, for these maximum and minimum points, uncertainty analysis will be done for longitudinal position (x) and wave height (z).

Each institution is encouraged to estimate their own precision limits, biases limits and uncertainty, and send it to the ITTC Resistance Committee.

When all data had been collected a Reference Value for Certification, obtained from all the institutions results, will be calculated and the final uncertainty analysis will be done by the Resistance Committee.

5.5 Data Submission

In a test series of this nature, it is important that an appropriate procedure is adopted in order to preserve the confidentiality of the data, and to avoid the possibility that the evaluation of the data could be affected by a knowledge of the facility from which it originated.

If the data were to be submitted by post directly to the evaluators, there are two obvious means by which data might be identified. Firstly, the stamp on the envelope, or the envelope itself, might indicate the country of origin. Secondly, the date on which the data were received might allow the participant to be identified from knowledge of the testing schedule.

In order to avoid this possibility, a two-stage submission procedure will be adopted involving a “moderator” to ensure anonymity of data. The moderator is not a member of the ITTC Resistance Committee. This procedure is described as follows:

The evaluators will send each participant an identical envelope marked only with the model size – i.e. 5.720m or 3.048m. When the participant has completed the tests, the data should be placed in this envelope, and sealed. The envelope should then be placed inside a second envelope and posted to the moderator. The moderator will open and discard the outer envelope only, and retain the sealed inner envelope. When the moderator has received submissions from all the participants, all of the sealed inner envelopes will be sent together to the Evaluators for analysis.

As a consequence of this procedure, the moderator sees only the outer envelopes, and does not see any data. Evaluators will not see the original postage stamps or outer envelopes, and will receive the data from all the participants together, eliminating the risk of identification described above. Thus neither moderator not evaluators will be able to identify the source of any of the data.

When the study is published participants will, of course, know from the figures which data is theirs.

In order to analyse the data easily, the results of the tests will be sent in electronic format using diskettes or a CD ROM. The diskettes or the CD ROM must not have labels or identifications that allow the committee to know their origin. Windows formats (as .xls from Microsoft Excel or .doc documents) that register the name of their creator must not be used; instead MS DOS raw data ASCII text files must be submitted with data ordered in columns separated using tabulations or commas. This kind of file does not allow the origin of the data to be determined. An example ASCII data file, in .csv format, will be distributed to all the participants, with instruction to generate it using Microsoft Excel.

5.6 Schedule

Twenty Institutions will participate in this worldwide series for identifying facility biases. Eleven of them will test each model.

The institutions have to inform the Resistance Committee about the reception and sending dates. The sender participant will cover the cost of the transportation of the models from one institute to another and the models have to come back to their origin.

Table 5.4- 5.720 metres length model.

Institution	Country	Month
CEHIPAR	Spain	Jun 2004
INSEAN	Italy	Sep 2004
Helsinki University of Technology	Finland	Nov 2004
ICEPRONAV S.A.	Romania	Feb 2005
Krylov Shipbuilding Research Institute	Russia	Apr 2005
Huazhong University of Science and Technology	China	Jun 2005
Samsung Ship Model Basin	Korea	Sep 2005
Akishima Laboratories	Japan	Nov 2005
NMRI	Japan	Feb 2006
NSWC	USA	Apr 2006
QinetiQ	UK	Jun 2006

Table 5.5- 3.048 metres length model.

Institution	Country	Month
CEHINAV	Spain	Feb 2005
LSMH/NTUA	Greece	Apr 2005
Inha University	Korea	Jun 2005
Universiti Teknologi Malaysia	Malaysia	Sep 2005
Australian Maritime College	Australia	Nov 2006
Canal de Experiencias de Arquitectura Naval	Argentina	Feb 2006
Stevens Institute of Technology	USA	Apr 2006
University of Glasgow and Strathclyde	UK	Jun 2006
Ecole Centrale de Nantes	France	Sep 2006
CEHIPAR	Spain	Nov 2006
INSEAN	Italy	Feb 2007

Each institution will have a month to test the model and send it to the next participant, with the schedule given in Tables 5.4 and 5.5.

5.7 Conclusions

In order to identify facilities biases a worldwide test series has been arranged using two geosims of a combatant hull. Twenty institutions are participating in the ITTC worldwide series for identifying facilities biases.

Test will finish at the beginning of 2007 but a preliminary uncertainty analysis will be done with the results of the first institutions in order to test the analysis procedures.

The dimensions of the model will be checked in the middle of the worldwide series.

A calculation sheet will be prepared to analyse the results of the tests.

5.8 Recommendations for Future Work

Continue tests of ITTC worldwide series for identifying facility biases.

Prepare a complete procedure for uncertainty analysis.

Analyse the results of the test in the middle and end of the worldwide series.

In the model definition the Committee found the necessity of a complete definition of model turbulence stimulation procedures, so this is recommended for future work.

6. UNCERTAINTY ANALYSIS IN CFD

6.1 Introduction

The 23rd ITTC RC recommended that users of CFD should rigorously implement best available approaches for Validation and Verification (V&V) of CFD simulations, as per QM Procedure 7.5-03-01-01 "Uncertainty Analysis in CFD, Verification and Validation Methodology and Procedures". It was also recommended that QM Procedure 7.5-03-02-01 "Uncertainty Analysis in CFD, Examples for Resistance and Flow" be updated as a collective example based on as many examples as possible of users of QM Procedure 7.5-03-01-01 for the Gothenburg 2000 Work-shop Test Cases. This section will attempt to cover what has been done in this area as well as the status of V&V for CFD in general.

6.2 Status of Uncertainty Analysis for CFD

QM Procedure 7.5-03-01-01 "Uncertainty Analysis in CFD, Verification and Validation

Methodology and Procedures" is based on the work of Stern and co-workers (Stern et al., 2001; Wilson et al., 2001, Wilson and Stern, 2002). It addresses the issue of computation verification for a particular problem primarily based on grid and iterative convergence. It does not address code verification, which is a separate issue. Code verification is often done by comparing calculations with simple problems for which exact analytical solutions exist. For a more rigorous code verification procedure Roache (2002) recommends the use of the Method of Manufactured Solutions (MMS). The MMS allows one to generate exact analytical solutions that can be used as benchmarks for verifying that codes used for solving partial differential equations are solving the equations correctly. To be applied to a particular code the code must be able to handle distributed source terms as well as general non-homogeneous time-dependent boundary conditions. However, once implemented the MMS can be used to help provide verification of the numerical accuracy of a particular code via grid convergence studies. The use of MMS to verify several aspects of an existing practical CFD code are demonstrated by Nelson and Roy (2004).

The ITTC Procedure for Calculation Verification was the recommended procedure to follow for the Gothenburg 2000 Workshop (G2K) where 7 of the 20 participants attempted to follow it. However, as reported by Larsson et al, (2003) there were problems with following the procedure due to the solutions being far from the asymptotic range and the lack of detailed experience with verification procedures for practical problems. Since G2K the ITTC Procedure has been applied to a number of practical problems.

One comprehensive effort for applying the procedure is that of Simonsen and Stern (2003a, 2003b) for the Esso Osaka. These tanker calculations include the rudder and are for straight ahead, static rudder, and pure drift conditions. The calculations provide uncertainty estimates for both the integral quantities as well as the flow field. Three systematically refined grids are used with the fine grid consisting of approximately 2 million points with a double body approximation for the free surface. Of particular note is that the rudder is included in the computation with a chimera grid showing the procedure can be used with these types of grids also. The authors express some doubt as to the validation of the computations due to the accuracy of the experimental data, but concentrate on the verification particularly for the flow field quantities. The uncertainty procedure has also been demonstrated for the DARPA SUBOFF submarine configuration by Van et al. (2003) who performed calculations for both the axi-symmetric bare hull configuration and the configuration with four stern appendages. The authors also computed the flow over the fully appended Coelacanth II, which is a 3000-ton class submarine hull form. To estimate uncertainty for the computed static pressure on the bare hull SUBOFF configuration the ITTC procedure is adopted with three different grids. The averaged L2 norm of interpolated values is used as a global metric and the authors claim the RANS solution is verified and validated at the level of 1.5% of the stagnation pressure coefficient. It is interesting to note that despite the authors' comparison with pressure velocity and force data for both the SUBOFF and Coelacanth II configurations they only perform the uncertainty analysis for the bare hull Cp.

The Recommended ITTC Uncertainty Procedure has also been applied to the unsteady case of a Wigley hull operating at forward speed in regular head waves by Weymouth et al. (2003). Here the procedure is used for the pitch and heave motions. Verification is done using three different grids as prescribed as well as different time steps. The time step study is done on the finest grid whereas the grid study is done with the medium time step. Validation is obtained for longitudinal force, heave and pitch motions on the finest grid with an average error of less than 2% when normalised by the maximum response.

For a more complicated geometry uncertainty estimates are obtained for the fully appended research vessel “Alliance” by Bull et al (2002). This computation includes propeller shafts and mounting struts as well as a rudder. Two separate calculations are performed, with different codes, apparently by different groups. One of the groups uses only two grids and comes up with uncertainty estimates. The other group uses three grids with every other point removed from the finest grid with a procedure that appears to be the ITTC Recommended Procedure.

The ITTC Procedure is similar to the Grid Convergence Index (GCI) approach of Roache (1998). A post-processing tool using Richardson extrapolation and the concept of GCI has been created by Cadafalch et al. (2002), which estimates the error band where the grid-independent solution is expected to be constrained as well as the order of accuracy of the numerical solution based on computations with three or more grids. Results for a number of flow computations and uncertainty estimates for various problems using GCI appear in the literature. Wilson and Stern (2002) show that the ITTC Procedure is equivalent to the GCI, but with a variable factor of safety (FS). This variable factor of safety is introduced by the correction factor in the ITTC Procedure, which increases with distance of solution from the asymptotic range. However, differences in details of the two methods as well as the interpretation of their results, and respective degrees of conservatism, are still under discussion. This is evidenced by recent articles in the literature about the two approaches (e.g. Roache, 2003 and Wilson et al., 2004). Both approaches estimate the uncertainty of the calculation by comparing solution differences on two grids of a similar nature for a particular problem. By using more than two grids an order of accuracy estimate of the scheme, as implemented for the problem, can be identified. The GCI uses a factor of safety based on how well the order of accuracy is established and how close it is to the theoretical value. With the ITTC Procedure a correction factor is used based on how the estimated order of accuracy compares to the theoretical. An important part of both approaches is the assumption that the grids used are of a similar nature and vary from each other in a regular way.

The Procedure recommended by the ITTC is not without controversy. As noted by Larsson et al. (2003), at G2K the procedure was not accepted by all participants and the need for improvement was stressed. As already discussed the GCI is an alternative to the ITTC Procedure. Another approach is that of Eça and Hoekstra (2002) who use a least squares approach. The least squares approach is used to deal with the scatter in the computed data. The scatter is often due to the lack of geometric similarity in the grids and the authors conclude it is almost unavoidable for complex flows around ships. They demonstrate their approach for six different computational problems and use a number of grids, larger than the minimum required. These cases included simple two-dimensional idealised problems as well as the flow over the Wigley hull and KVLCC2 tanker. Typically, the order of accuracy of the scheme as implemented for a particular problem using a minimum of four grids. The extra grids are deemed necessary by the authors to better handle the effects of the scatter or one has to choose more conservative values for the uncertainty.

One of the biggest drawbacks of the above methods is that they have all been formulated for regular structured grids and rely on Richardson extrapolation for quantifying the errors. There is a fundamental requirement that the grids used to estimate the uncertainty be of a similar nature. Three grids, all within the asymptotic range, are needed to determine the order of accuracy of the calculation. A constant grid refinement ration is also used implicitly in the derivation. Such requirements are difficult to achieve for complicated problems. Indeed, Eça and Hoekstra (2002) conclude that it is not yet feasible to attain the asymptotic range for ship hydrodynamics problems so it is difficult to establish the order of accuracy. These requirements may be even more impossible to achieve with unstructured grids. Applying the above methods to unstructured grids require the use of an "effective grid refinement ratio" which is often ad hoc. Also, the grids that are being more commonly used for addressing complex problems involve combinations of grids, which can include mixed elements, chimera, overset, and local refinement as possible options. Consequently, the methods being used today for uncertainty analysis of CFD solutions may have limited use with the way CFD codes are evolving.

Ultimately it would seem the use of adaptive grids to minimise grid dependence error is the ideal solution to estimating grid induced error. Such an approach is taken by Pelletier et al. (2003). By using unstructured adaptive grids the mesh refinement process can be automated and numerical errors controlled. However, for the problems demonstrated the authors can only show the trend toward grid convergence, but cannot necessarily achieve it due to computer limitations. Solutions are obtained on various grids due to the grid adaptation process, but again they are not changing by constant amounts so the above methods are not directly applicable. An interesting facet of uncertainty estimates in computations that is often ignored is that of parameters such as viscosity and measured temperatures. Pelletier et al. (2003) demonstrate flow sensitivities due to uncertainties in such model parameters.

The use of error transport equations (ETEs) may be an option that can address the complicated computational grids that will probably become more common in the future. The use of a transport equation for the error, which can be solved on the same grid as the computation is very attractive as it can be done as part of the computation or as a part of post-processing. It also starts to account for the very real physics that errors once generated can be transported to other parts of the flow field. The main issue with ETEs is the derivation of the residual or error source term in the transport equation. A number of approaches have been proposed in the literature (e.g. Zhang et al., 2001; Celik et al., 2003a, 2004; Qin and Shih, 2003) and this is an ongoing area of study. Unfortunately, ETE approaches have largely been for simple model problems to date and do not necessarily provide convergence information. However, Qin et al. (2004) have applied their approach to the Euler equations on unstructured meshes that can be triangular, rectangular, or other polygons indicating the possible generality of the method once it matures.

The concept of calculation validation for a LES becomes even more complex. With LES only turbulence levels at the smallest scales are modelled with all other scales directly resolved. As one refines a LES grid it will tend to resolve more of the flow field and model less of it. Consequently, it may be impossible to obtain a grid independent LES as it will tend to become a DNS of the flow field on the finer grids. However, this is not to say some level of quality assessment of the LES solutions should not be done. A start toward this is that of Celik et al. (2003b) who propose various index of quality measures for LES computations and apply them to some case studies.

Unfortunately, there does not appear to be a large acceptance of using any of the above procedures for estimating uncertainty of CFD solutions. At the recent 25th Symposium on Naval Hydrodynamics, held in St. John's, Canada, and the 7th Numerical Towing Tank Symposium, held in Hamburg, Germany, where a significant number of CFD related papers have been presented, almost no one performed an uncertainty analysis. This is not to say that uncertainty estimates for CFD are not important. In fact, in October, 2004 a "Workshop on CFD Uncertainty Analysis" was held in Lisbon, Portugal. Two flow problems including: flow over a hill and flow over a backward facing step, were evaluated by six groups. Single block, geometrically similar structured grids were distributed by the organisers to the participants. Three uncertainty estimation methods were used by the participants including: the GCI with factor of safety of 1.25, the Eça-Hoekstra least-squares approach, and an ETE method. According to Roache (2004b) there was a clear consensus that the results were consistent and provided a favourable evaluation of all three approaches to uncertainty estimation.

6.3 Conclusions

At the recent "CFD Workshop Tokyo, 2005" there was a concerted effort by many of the participants to apply uncertainty analysis to their computations. Although there is an evident lack of consensus on this topic there is growing effort and attention being given to it, which is clearly a step in the right direction for computations. Prior to the workshop, the ITTC procedure for estimating the uncertainty of a particular solution, or any other procedure for that matter, did not appear to be commonly used in practice. This is despite the fact that grid studies are now more commonly done. The results of the "Workshop on CFD Uncertainty Analysis" carried out in Lisbon 2004 seem to indicate there is no clear consensus on the correct or best procedure to use for two-dimensional problems on structured grids let alone complicated practical problems or unstructured grids. This may indicate the state of the art and the possibility that it may be premature to impose a procedure on the community at this point. Part of the solution may be the imposition of pressure on code builders to include better methodologies in the code directly to help perform verification and validation for a particular solution. Part of the problem is also probably that many efforts are aimed at dealing with more complicated geometries and complicated physics requiring evolving grid technologies including: grid adaptation, unstructured, mixed elements, chimera, local refinements and other methods to treat practical problems. The uncertainty estimation procedure subscribed by ITTC and those of the CFD community in general have not necessarily kept pace with these evolving complexities. However, the use of uncertainty estimates for CFD studies needs to be encouraged. To this end it is recommended that the ITTC maintain its current recommended procedure on an interim basis, but also encourage the use of other techniques if so desired by particular users.

7. FAR FIELD WAVES AND WASH

7.1 Introduction

The subject of ship waves and their impact on the marine, coastal, and river environments has attracted increasing attention in recent years. Concerns have been raised over erosion, damage to wildlife habitats, damage to coastal engineering structures such as sea walls, damage or nuisance to moored vessels, and danger to small craft at sea and to people working or enjoying leisure activities near to the water's edge. Several serious accidents, including some fatalities, have been blamed at least partially on wash waves from ferries (see for example MAIB report No 5/2004).

In many cases, problems have been blamed on the introduction of fast ferries on routes previously operated by conventional ferries; however other problems have also been related to increasing speed or traffic density of conventional ships. The prediction of wash effects has thus assumed increasing importance in the planning and operation of ferry services as well as in the design of vessels.

The prediction of far-field wash waves in realistic situations is a challenging problem, since the wave properties are often required at great distances from the ship, in many cases in regions where the bathymetry varies in a complex manner relative to the track of the ship. This leads to a need to solve unsteady problems in large computational domains, in which wave transformations due to refraction or diffraction can be of great importance. In many cases hybrid solutions have been adopted in which a near-field model of the ship wave generation based on an assumption of uniform water depth is coupled to a far-field model. This can be for either of two reasons; firstly to extend the calculations domain for a non-linear near-field approach using a more computationally efficient linearised far-field approach, or to couple a ship-wave model based on uniform depth to a model representing the wave transformation due to non-uniform water depth in the far field.

Ship wave patterns in finite depth water are commonly divided into three categories: sub-critical, in which the Froude number based on depth, $Fr_h < 0.9$, trans-critical (or critical), in which $0.9 < Fr_h < 1.1$, and supercritical, in which $Fr_h > 1.1$, though the boundaries between these regions are not precise.

In some cases the problems are not confined to the Kelvin wave pattern. Some environmental effects, especially when large ships are operating in restricted waters have been attributed to Bernoulli waves, also known as "drawdown".

Whilst there have certainly been developments in the prediction of wash waves from numerical techniques, the subject is still evolving; some promising numerical methods exist, but in many cases lack of appropriate physical data has prevented wide-ranging validation. It is thus considered still too early to propose guidelines for wash wave prediction.

7.2 Prediction of Wash Waves Based on Experimental Measurement

Challenges. There are a number of challenges associated with the prediction of wash waves based on model test data. Restrictions on the width of tanks often result in the need to make measurements relatively close to the model track; however extrapolation to the far-field, even assuming uniform water depth, is usually based on an assumption of linear theory and far-field properties. Wave elevations must thus be measured sufficiently far from the model that near-field effects are negligible; the required distance scales with both model length and *Fr* squared. Measurements must be unaffected by reflections at the tank wall, unless they are explicitly accounted for in the analysis. In many practical cases, the water depth must be scaled correctly in order to represent the depth Froude number correctly. In many cases it will be hard to meet these requirements.

Particular issues arise if the speed range is trans-critical, when the depth Froude number is close to one. Several authors (see for example Jiang, 2001) report that solitons may be generated upstream of the ship, whilst others (e.g. Doyle et al., 2001) report a growth in wave height as the model progresses down the tank.

It is commonly argued (e.g. Molland et. al., 2004, Macfarlane and Renilson, 2000) that the properties of the wave envelope, such as the maximum wave height, can be extrapolated from one transverse location to another in uniformly deep water using a simple exponential decay rate calculated for a point disturbance. The decay exponent for a point disturbance with distance is –1/2 within the Kelvin angle, and –1/3 for a wave cut along the extremities of the Kelvin wedge; many authors thus argue that the divergent waves generated by a ship in deep water decay with an exponent of –1/3; other authors, however, contest the validity of the application of this result to wave patterns generated by ships rather than point disturbances (e.g. Doctors and Zilman, 2004, Doctors and Day, 2001). The result is certainly not valid in water of finite depth, in which the decay rate can be reduced. Decay exponents between –0.2 and –0.5 have been calculated from numerical data with different ships and in different conditions of speed and water depth.

A preferable approach is to derive the far-field wave spectrum using a wave-cut technique, subject to the usual limitations associated with these techniques. The wave elevation can then be found anywhere in the far field by a straightforward calculation.

All of these challenges affect in turn the ability to validate CFD predictions. In many cases towing tank data is used for validation, which is generally gathered closer than desirable to the vessel track for validation of a far-field prediction. Little or no data from far-field full-scale measurements has been published in a form which would allow CFD validation; that is, including vessel details as well as wave data.

Recent Work. Molland et al. (2004) carried out a series of tests over a range of speeds corresponding to length Froude numbers between 0.25-1.0, and depth Froude numbers between 0.5-2.0. Results show maximum wave heights for four slender ship forms each tested in four configurations: monohulls (consisting of a single demi-hull) and catamarans with various demi-hull spacing.

A further set of tests was carried out for one configuration to examine the effect of the propulsion method. Three different methods were employed: a conventional towing test, self-propulsion by propeller, and by water-jets. Results show that the propulsion method had relatively little impact on the leading waves in the cut, though the discrepancies were slightly larger after several waves had passed.

A final set of tests investigated the influence of model scale. One hullform was constructed at two scales, corresponding to lengths of 1.6m and 4.5m. The results indicate that the wave cuts are quite similar for the leading waves; no real conclusions are possible for the later waves due to the different influence of reflections in the two tanks used.

Allenström et al. (2003) carried out an extensive series of measurements of the influence of a vessel passing an enclosed rectangular bay, bounded by a vertical wall. Particular attention was paid to the amplification of the waves inside the bay. Tests were carried out over a range of speeds, different distances from the "coastline", and at two uniform water depths, using a 1:36.9 scale self-propelled model of a 194m ROPAX ship. The authors concluded that for a relatively large ship such as a ROPAX vessel, operating in relatively shallow water, the influence of the Bernoulli wave (or drawdown) may create more problems than the Kelvin-like ship wave pattern. The water depth was found to play a key role in the amplification of the Bernoulli waves; lateral restrictions, such as banks, were found to increase the amplification further. The Bernoulli wave was found to radiate a long distance transversely in shallow water.

Results show that when the ship passed close to the entrance of the bay, the drawdown was significantly larger in the bay than at the bay entrance. Kelvin waves were found to be largest in the innermost part of the bay, in regions in which reflections of the divergent wave system from the vertical wall interfered constructively with further divergent waves.

7.3 CFD Prediction

Linear Theory. A number of approaches are possible for the calculation of ship-wave patterns. At the simplest level, linearised theories based on Kelvin singularities may be used, and have a significant advantage over most alternatives in that calculations in the far-field can be implemented without the need for calculation of the intervening wave-pattern, as long as the water depth can be considered uniform. Many such approaches are based on either the slender body or thin-ship approximations; for typical high-speed ships these approaches require a heuristic representation of the flow from the transom stern, and typically do not explicitly model dynamic sinkage and trim effects.

In many cases (e.g. Molland et al., 2003), notwithstanding the reservations mentioned above, the leading part of the wave cut is predicted well by the linear theory as long as the ship hull or demihulls can reasonably be considered slender. For vessels with several slender hulls, such as trimarans, linear theory is seen correctly to predict trends in wave effects due to hull spacing or stagger (Doctors and Zilman, 2004b). Near-field waves can also be calculated if required (e.g. Day and Doctors, 2001).

A related theory allows the prediction of wash waves from air-cushion vehicles or planing ships (e.g. Doctors and Day, 2000, Tuck et al., 2002 and Doctors, 2004); the theory can be combined with thin ship theory to allow the prediction of wash waves from surface effect ships.

A number of authors have explored ways in which some real fluid effects can be incorporated into linear theory. Tuck et al., (2000, 2002) extended the thin-ship theory to include the effect of eddy viscosity in deep water; they argue that the wave patterns thus generated are more realistic, particularly in the region close to the vessel track, due to the absence of some of the fine detail predicted by inviscid theory which is not observed in practice. Doctors (2003) extended this approach to finite depth.

Following a study by Zilman and Miloh (2001) of the effect of surfactants on Synthetic Aperture Radar (SAR) ship wake images, Doctors and Zilman (2004a, 2004b) extend the theory further to include the effects of surface elasticity as well as eddy viscosity in finite depth water. However the calculated wave cuts show that the resulting effects are generally very small.

Non-linear theory: Uniform Water Depth, Steady Flow. In order to avoid the approximations required for linear theory, it is possible to adopt a hybrid approach, utilizing a non-linear flow approach in the near field, matched to a linearised approach in the far field. This approach relies on the assumptions that the waves are less steep in the far-field, and can reasonably be considered to satisfy the linearised boundary conditions.

In most cases the near-field model is based on non-linear potential-flow theory, typically using a Rankine panel approach. When Rankine sources are used to model the free surface, the near-field wave pattern is only predicted in the panelled region; this must extend far enough from the ship to avoid the near field effects. Raven (2000) emphasizes the importance of minimising numerical damping in the non-linear solution in order that the extrapolation to the far field is independent of the location of the matching region. Raven's study includes near-field calculations in uniform finite water depth, in which the hull and free-surface panel distribution is mirrored using the flat sea bottom as a symmetry plane. Good agreement with experiments was found for the leading waves in longitudinal wave cuts close to the ship.

For the case of uniform water depth, and sub-critical speed, the results are extrapolated to the far field by carrying out a wave pattern analysis using a transverse cut approach. The far far-field wave pattern is then reconstructed from the spectrum. A comparison is made between predictions for a longitudinal wave cut located 1.125 ship lengths from the track using a direct non-linear calculation over the whole domain, and an extrapolation using the far-field spectrum from a near-field calculation over a smaller domain with coarser panelling. Good agreement is demonstrated for the few waves shown.

Raven (2000) comments that for trans-critical speeds there is little point extrapolating the near-field waves to the far field due to the lack of validation of the near-field solution in these conditions. For supercritical speeds he suggests one possibility is to adopt a conservative assumption of no decay, due to the non-dispersive nature of the outer waves bounding the wave pattern; however he cautions that some waves in the pattern will still be sub-critical and will decay in the usual manner due to dispersion.

Brizzolara and Bruzzone (2003) adopt a broadly similar approach using a non-linear Rankine panel method to model the near-field, and extrapolating into the far-field using a free wave spectrum derived from transverse cuts. Only limited results are presented; unusually amongst these studies, the agreement between the numerical prediction and experimental measurement is least good for the leading wave, and better for the subsequent waves.

Janson et al. (2003) also adopt a hybrid approach based on a non-linear Rankine source method in the near field. In their study, Rankine sources are distributed over the hull, the inner domain free-surface and a vertical "matching wall". Far-field extrapolation is performed by distributing Kelvin sources on the matching wall in order to specify the boundary condition for the disturbance velocity potential and to compute the far-field waves. Calculations are carried out for deep water only.

Numerical tests illustrate the importance of locating the matching wall sufficiently far from the ship, and compare solutions obtained using the extrapolation technique with those using the non-linear calculation over a larger domain, both for wave cuts inside and outside the matching wall. Comparisons with experiments are also presented, indicating good agreement over the leading waves in the cut, but less good agreement for the subsequent waves.

In contrast to the previous methods employing potential flow techniques to solve the near field, Guillerm and Alessandrini (2003) present a coupled method in which RANS equations are solved for the viscous flow field near the ship while the outer domain is solved using a potential flow approach, using the Fourier-Kochin formulation. The two regions are strongly coupled through the velocity, pressure and free-surface elevation. This procedure provides a better prediction of far-field waves than would be obtainable with RANS alone since the potential-flow computation avoids the excessive numerical damping of the RANS, whilst also allowing the potential for larger computational domains.

Non-linear Theory: Non-Uniform Water Depth and Unsteady Flow. In many practical cases of interest, the wash waves propagate into areas where the depth is non-uniform. If the ship is travelling at constant speed along a channel of arbitrary uniform cross section, the problem is steady, and can be approached using a conventional non-linear approach, such as a Rankine panel method, as described above, as long as the size of the channel is sufficiently small. Raven (2000) describes non-linear calculations for a ship travelling parallel to a uniform sloping bank bounded by a vertical wall; in this case the bottom and wall were panelled. Comparisons with experiments indicate reasonable agreement; however the reflection is not completely resolved, which leads to underestimation of wave heights at the wall.

In practice, many wash problems are affected by local topography, which cannot be represented as steady within a ship-fixed reference frame. Problems of interest include shoaling, refraction, reflection, focusing, and penetration into harbours. In such cases, the approach generally adopted involves a one-way coupling between a near-field approach to model the ship-wave generation, and a far field approach which models the wave evolution. The significance of the one-way nature of the coupling is that no account can be taken of the effect of phenomena from the far-field within the near field. Both linear and non-linear models have been adopted for the near field solution; most recent studies use Boussinesq-type models in the far field, although other approaches have been adopted (e.g. Kofoed-Hansen et al., 1999 and Whittaker et al., 1999).

Raven (2000) describes the coupling of a inner model based on non-linear Rankine panel method for ship-wave generation to an outer model based on non-linear Boussinesq-type equations. The outer model accurately represents wave dispersion and shoaling for waves longer than about twice the water depth.

Yang (2002) examines a variety of approaches based on Boussinesq-type equations, with two different methods for generating the ship waves. One method uses thin-ship theory to represent the ship-wave generation, and couples the results for free-surface elevation and horizontal velocities along a matching boundary. In this approach, there is an inconsistency between the linear thin-ship theory and the non-linear free surface condition in the Boussinesq equations; moreover the flow must be steady. In the alternative approach the body boundary condition on the centre-plane of the ship is taken from slender body theory, implying small or moderate Froude number. A numerical study of the generation of upstream solitons at critical speed is presented; comparisons are made with experiments, showing good agreement for measured wave profiles.

Jiang et al. (2002) use a slender-body theory to approximate the near-field flow coupled to Boussinesq-type equations. A range of numerical studies are presented, including prediction of the waves generated by ships in channels of different cross section shape, of the effect of different acceleration patterns for a container ship moving over an irregular near-shore fairway, and of the evolution of the wave pattern of a fast ferry moving over a ramp designed to minimize the time spent at trans-critical speeds. Comparisons are made with data measured in a towing tank; agreement is good close to the sidewalls of the tank and ahead of the model. A second set of comparisons predicts the progress of a harmonic wave train over a two-dimensional bar. Results agree well with experiments up until the point at which the higher harmonics generated by non-linear effects become free as the water depth rapidly increases on the down-wave side of the bar. This lack of agreement is attributed to the need for a correction to the dispersion relation within the Boussinesq equations.

A variation on this approach is presented by Hänel et al. (2003) who solve the near field using a three-dimensional finite volume solver, coupled to a shallow water Boussinesq method in reduced dimensions in the far field. This allows the flow features in the vicinity of the ship to be predicted with RANS and the far field shallow water effect to be computed with much more efficiency than possible with RANS for a given resolution of the waves.

In contrast to the methods employing Boussinesq equations, Aelbracht et al. (2000) use a spectral approach in the far field to model propagation of waves into a complex bathymetry. The directional energy spectrum is obtained explicitly in terms of the free wave spectrum, determined from a deep water quasi-steady solution of a Neumann-Kelvin model. Numerical results only are presented in terms of the evolution of significant wave height with time for variations of vessel track and speed.

7.4 Conclusions

The solution of real-world wash wave problems, with the combination of extremely large computational domains, complex bathymetry and unsteady effects remains highly challenging.

Significant progress is being made in numerical approaches for the prediction of far-field waves, with hybrid methods showing great promise. However, the routine solution of real-world problems is still some way off. Experiment studies, in contrast, are highly constrained by problems of size and scale, and the difficulties of simulating realistic bathymetry; nonetheless some interesting and important phenomena have been reported. At this time the RC does not believe that enough experience exists to propose general guidelines for the prediction of far-field waves and wash effects.

To some extent the credibility of the numerical approaches is compromised by the lack of appropriate public-domain experiment data for benchmarking purposes. It would be highly desirable for researchers to have access to data appropriate for comparison purposes, ideally over a realistic range of speeds, water depths, and distances from the vessel track for a variety of vessel types and over both simple and complex bathymetry.

8. DEVELOPMENTS IN MODELLING OF RELEVANCE TO RESISTANCE

8.1 Introduction

In this chapter, recent developments in modelling of relevance to resistance are described. Most models are either used in association with Computational Fluid Dynamics, or arise from Experimental Fluid Dynamics.

8.2 Waves and Free-Surface Effects

Breaking Waves. Breaking waves may be treated by a variety of methods as MAC (Marker and Cell), VOF (Volume of Fluid), Level Set (single phase and two phase), density function, and recently SPH (Smooth Particle Hydrodynamics) by Colagrossi et al. (2004), and CIP (Constraint Interpolation Scheme) by Hu et al. (2004).

Muscari and Di Mascio (2003a) developed a spilling breaking model for wave flows around ships. This model is based on the assumption that the breaker can be approximated by an eddy riding the forward face of the breaking wave. An algorithm for the detection of the breaking onset and the evaluation of the breakers extension is given. Only the pressure term is taken into account. Viscosity effects are neglected. The detection of breaking zones and wave amplitude damping are reproduced very effectively. However, the onset of the turbulent trailing wake is not captured. The viscosity effect resulting in a slight reduction of the wave length of the following wave train is taken into account in Muscari and Di Mascio (2003 b).

Free Surface Boundary Conditions. Iafrati and Campana (2004) investigated small scale breakers generated by a submerged hydrofoil. The two-dimensional unsteady Navier Stokes equations for the two-phase flow of air and water are solved. The surface tension is modelled as a continuum force.

Doctors and Zilman (2004) investigated the influence of surface tension and viscosity on the wave-making of a model catamaran. The influence of surface tension and the viscosity is taken into account in the free-surface condition of the perturbation velocity potential. The obtained solutions are potential flow solutions. The inclusion of the free-surface tension in the theory improves the accuracy of the results. It is also shown that the surface tension is important at low Froude numbers when the model is as small as the one investigated.

The role of surface tension in the modelling of ship waves was investigated also by Chen (2002). A steady free-surface potential flow generated by a source advancing at constant speed is considered. The inclusion of the surface tension in the free-surface boundary condition gives a more realistic description of ship waves, and the singularity of the Green function when the source and field points are located at the free surface will be eliminated.

Doctors (2003) investigated the influence of viscosity on the wave-making of a catamaran model. The turbulent viscosity is taken into account in the respective free-surface boundary condition of the velocity potential. The inclusion of the turbulent viscosity improves the prediction at high Froude numbers and gives a closer correlation between the theory and the experiment at the lower Froude numbers.

Schweighofer (2003) performed a very comprehensive investigation on two-dimensional transom waves using different free-surface boundary conditions at model- and full-scale Reynolds numbers. The Euler and RANS equations are solved and the free-surface boundary conditions are realised in numerous ways. The influence of the turbulent viscosity on the wave-making is negligible for high Froude numbers at model- and full-scale ship Reynolds numbers. But at lower Froude numbers a slight influence is observed. The way in which the free-surface boundary conditions are realised has a remarkable influence on the computed wavemaking, which is traced to the different numerical damping appearing.

Transom-Stern Flow. When solving the potential flow, the transom is usually treated as dry. The boundary conditions at the transom are either obtained from the Bernoulli equation where the pressure is set to the atmospheric pressure being equal to zero, Saisto (2002), or a fictitious pressure is defined being dependent on the Froude number based on the transom depth, Kyoung et al. (2003).

Solving the RANS or Euler equations, the transom may be treated as either dry or wetted either by setting the wave height there equal to the transom immersion, Schweighofer (2003), by limiting its movement upwards to the transom contour, Li (2002), or by evaluating the wave height there by extrapolation from the computational domain, Li (2002).

Yamano et al. (2001) investigated the scale effect on the stern-wave resistance presenting results up to full-scale, using empirical relations for the velocity and boundary layer thickness at the stern, whereby the waves and the flow lines below the waves were approximated by trochoids.

8.3 Viscous Flow

Stern Longitudinal Vortices. The application of RANS methods to the evaluation of stern longitudinal vortices is basically possible. Nevertheless, their application is always associated with numerical damping to a certain degree. The appearing vortices may be dampened out, and the evaluation of the wake may not reveal the influence of the vortices on the local flow field. The computational times may be increased due to an increased amount of necessary computational cells.

An alternative to the application of RANS solvers to such problems is given by the vorticity confinement approach, Dietz et al. 2004), Fan and Steinhoff (2004), Fan et al. (2002), Steinhoff et al. (2003b) and Steinhoff et al. (2003a). The basic idea of the vorticity confinement approach is to solve the continuity and momentum equations with added terms controlling the size and time scales of the vortical regions or vortical boundary layers. These terms consist of one term related to the numerical diffusion and another term related to the vorticity which is regarded as an anti-diffusion term compensating the appearing numerical diffusion. The terms are activated only in the vortical regions of the flow. Anywhere else they will vanish. For far less computational cost, the obtained results were at least as good or better than the ones obtained using conventional methods. The results were related to basic flow problems as flows around blunt bodies, spheres, cylinders and airfoils. Lohner et al. (2002) have applied the technique directly to the RANS equations for the purpose of preventing the dissipation of computed vortices due to inadequate grid resolution. With RANS computations and their highly resolved boundary layer grids it is necessary to add a switch to help turn off the term based on the local Reynolds number. Results are demonstrated for a NACA wing as well as a submarine with sail planes.

Roughness. A good survey of the literature with respect to models for roughness is provided by Raupauch et al. (1991). Another source is provided by Patel (1998).

Solving the RaNS equations, Menter's BSL k-ω turbulence model, Menter (1993), or a two-layer k-ε turbulence model, Durbin (2001), may be used. In the first turbulence model, according to Wilcox (1993), the hull roughness is taken into account by a modified boundary condition for ω where the sand-grain roughness is included resulting in a limited value for ω instead of infinity. The latter one modifies the smooth-wall formulation by introducing a hydrodynamic roughness length into the appearing equations, and modifying the boundary condition for k, which becomes non-zero at the wall. Using experimental flat-plate results, the turbulence model is calibrated where the hydrodynamic roughness is related to the true, geometric roughness.

Kotey et al. (2003) propose a novel modification to the power-law velocity profile to account for the effect of surface roughness in the overlap region. The introduced roughness parameters reveal more accurately the effect of roughness on the skin friction than the shift of the log-law. Further, the non-dimensional velocity profile is reproduced better than by the logarithmic relation.

Leer-Anderson and Larsson (2003) propose a combination of experimental measurements with a numerical method for the evaluation of the skin friction of full-scale ships with surface roughness. The roughness effect is represented as a downshift in the logarithmic part of the boundary-layer velocity profile. This shift is linked to a formula for the local skin friction, and integration of the local skin friction over the hull will give the total skin friction. In the formulation for the velocity-shift function appears a relation between the roughness height and an empirical coefficient which is obtained by pipe-flow experiments for the ship to be evaluated. The friction velocity is approximately the same as on the ship. The proposed approach seems to be of greater accuracy than empirical methods, and it shows that large differences in skin friction may be obtained for different surfaces with the same average roughness height.

Castillo et al. (2004) performed two-dimensional turbulent boundary-layer experiments with zero pressure gradient. The measurements are unique as the achieved Reynolds numbers are very high (Re_θ about 1.2×10^5). They report that the effects of the upstream conditions, the Reynolds number and the roughness are completely removed from the outer flow when the mean deficit profiles are normalised by the Zagarola/Smits scaling. The true asymptotic profile in the turbulent boundary layer is found in zero-pressure-gradient flow regardless of the range of the Reynolds number, the surface conditions and the initial conditions. This finding might be applied to the development of surface-roughness models in the future.

Subramanian et al. (2004) indicate in an experimental study that algae roughness or roughness by bio film cannot be treated in the same manner as roughness caused by rigid micro-objects. The inner-law (log-law) dynamics may be abnormal compared with the one related to rigid micro-objects. The assumption of a negligible normal pressure gradient may no longer hold for zero-pressure-gradient flows. The Clauser-velocity-loss function should include a form drag factor additionally to the viscous drag factor.

8.4 Turbulence

Transition. Some progress has been made in the development of general purpose models for the prediction of transition in the framework of the RANS equations. However, it remains to be seen if these models are appropriate for the prediction of ship resistance. A comprehensive account of the state-of-the-art in turbulence and transition is provided by Launder and Sandham (2002)

Menter et al. (2004) and Langtry et al. (2004 and 2005) have developed a correlation based model that is built strictly on local variables and uses transport equations for the intermittency and the momentum thickness Reynolds number.

8.5 Drag Reduction

In the last years, the environmental requirements and governmental regulations related to vehicle technology have become stricter (e.g. Kyoto treaty). Environmentally friendly technical solutions and a reduction of polluted exhaust gases are requested. Additionally, the competition has increased in the field of maritime technology requiring more economical vessels. Therefore, the issue of drag reduction has become more important.

The wave resistance of a ship may be reduced by model testing or the application of CFD very efficiently.

The frictional resistance of a ship may be reduced by the following means: injection of micro bubbles, air films and polymers, highly water repellent or super water repellent coatings, magneto hydrodynamics and surface shaping as in the application of the tornado-like technology, TLT, The Royal Institution of Naval Architects (2002). Most common and also most investigated seems to be the use of micro bubbles and polymers. However, full-scale results are very rare. Kodama et al. (2002) reviewed and analysed measurements on the skin-friction reduction effect by micro bubbles discussing the mechanism of the skin-friction reduction including the effects of density, bubble size and reduction of Reynolds stresses. A full-scale micro-bubble experiment was conducted using a 116m long ship. The resistance was reduced by three percent giving a net power saving of two percent.

Recently, studies related to the reduction of the skin friction of ships, experimental and computational ones, have been performed in the USA, Germany, the Netherlands and Japan.

In the USA, the Defense Advanced Research Projects Agency (DARPA) initiated the Friction Drag Reduction Advanced Technology Programme where multi-scale modelling capabilities are developed allowing researchers to run full-scale experiments on a computer. A list of impressive publications may be found at:

http://www.darpa.mil/ato/programs/friction.htm.

In Germany, at the Hamburg Ship Model Basin (HSVA), research related to the Tornado-Like-Technology (TLT) is carried out. The Tornado-Like-Technology (TLT) is based on the modification of the flat surface by spherical flat recesses (dimples) creating a secondary flow having an influence on the boundary-layer flow. The modified boundary-layer flow will result in a lower skin-friction resistance.

The Maritime Research Institute of Netherlands (MARIN) is leading a joint industry project which is focused on the application of air lubrication and super water repellent coatings to skin friction reduction.

In Japan, respective research was started in the nineteen nineties at the University of Tokyo being partly joint with the research at the Ishikawajima-Harima Heavy Industries Co. (IHI). In the National Maritime Research Institute Japan (NMRI), research with respect to the reduction of the frictional resistance of ships was started in the end of the nineteen nineties. The respective references may be found in Kodama et al. (2002).

Apart from ship hydrodynamics a vast amount of references with respect to numerical and experimental investigations of the reduction of skin friction may be found from other fields, such as pipe flow and nuclear technology.

Some results of the programmes mentioned above, but also of numerous other research institutions, will be presented at the 2nd International Symposium on Seawater Drag Reduction to be held in Pusan, in Korea, in 2005. This symposium seems to be a good starting point for the review to be performed in the 25th ITTC Resistance Committee.

8.6 Conclusions

Most models of relevance to resistance are used in association with CFD, particularly with the solution of the RANS equations. This makes it difficult to distinguish between the development of pure models and CFD. Many physical phenomena such as free surface deformation, longitudinal vortices, roughness, turbulence and physics of drag reduction may be taken into account in the respective evaluations. However, the application of some of the presented models seems to be limited to a few cases being sometimes relatively simple. A general application of most of the presented models to flows around ships on a daily basis is not yet observed. As the physical phenomena presented have a certain significance in the resistance estimation of ships, and as direct numerical simulations of ship flows cannot be applied in the foreseeable future, the development of respective models being more generally applicable will continue. The significance of drag reduction will certainly increase due to the stricter regulations related to the environmental requirements of different countries, whilst improved competition will also require new means and evaluation techniques related to drag reduction. With respect to drag reduction, a vast amount of investigation has been carried out - too much to allow a reasonable overview of the different models developed to be given here.

It is recommended to continue the monitoring of developments in modelling of relevance to resistance, particularly with respect to the general application to ship flows. Further, it is recommended to devote a separate task of the 25th Resistance Committee to the drag reduction of ships.

9. NEW DEVELOPMENTS IN SHIP CONCEPTS, DESIGN METHODS AND DESIGN OPTIMISATION

Relevant political and economical factors, such as the efficiency and economy of shipping with respect to land and air transport or the great environmental compatibility of sea transports, seems to support the growth of the fleets. The central role of shipping is clearly underlined by the fact that about 90% of the world trade is being transported by ship. In this scenario, the need for new designs with increased safety, efficiency, flexibility of use, speed and comfort, is becoming evident.

9.1 New Ship Concepts

Whilst far from being the only factor driving innovation, safety is however a major element of public concern, especially when pollution of the environment is involved, and the quest for designs capable of reducing the risks of shipping is challenging design engineers.

In reviewing some current trends in ship design, Payer (2004) stresses the following challenges for different ship types: tankers, for which redundancy in propulsion and steering is becoming an issue; bulk carriers, with an increasing demand of more efficiency; passengers ships and ferries of constantly increasing speed and length.

Among the recent papers dedicated to the development of novel ship concepts appeared in literature, Mewis and Klug (2004) analysed current trends in container ship design and illustrate alternative concepts, dealing with the problems posed by the constant increasing of their capacity. Design challenges posed by Compressed Natural Gas carriers are illustrated in Valsgard et al. (2004). Eefsen et al. (2004) report the summary of a European MoD research project (THALES). In the study, alternative designs have been produced and resistance and seakeeping performances of seven different concepts has been investigated and compared, including tumblehome, enlarged, axe bow, wave piercing and COFEA (Centre of Flotation Extreme Aft) concepts.

A broad review of some of the most promising new designs for fast ships is presented in Bertram and Seif (2004), following a classification of the marine vehicles based on hydrostatic buoyancy, hydrodynamic lift and powered lift. A wide variety of designs is presented, including multihulls, pure and hybrid hydrofoils, cushion vehicles and lift vehicles.

Among high-speed multihull solutions, catamarans are nowadays the most popular concepts. A discussion on speed limits of hydrofoil and air-cushion (SES) catamarans posed by foils cavitation and by resonance oscillations of the air-cushion (“cobblestone” effect) is given in Faltinsen et al. (2004). A new concept of Air-Assisted Vessel (AAV) is developed as a part of a European Project EFFISES.

Displacement trimarans are a more recent development compared to catamarans. Their advantages with respect to monohulls are lower resistance in the high speed range, enhanced survivability and enhanced seakeeping characteristics. The interest in this concept (sometimes referred as stabilised-monohull) is developing at a fast pace, generating a number of new design studies, based on both numerical simulations and model testing (Armstrong and Holden, 2004, Brizzolara et al., 2003, Day et al., 2003, Barone and Bertorello, 2004, Floden et al., 2004). A large ocean-going demonstrator, the RV Triton has also been launched and an extensive series of trials was carried out to measure her hydrodynamic performances (Renilson et al., 2004), assessing the benefits of this configuration over conventional monohulls. A recent RINA conference (2004) was entirely dedicated to the design and operation of trimaran ships. Armstrong and Holden (2004) presented an interesting paper with an outline of the design philosophy of a trimaran and a description of the design process.

9.2 CFD in Ship Design

The combined use of mature CFD solvers and design techniques has received an increasing attention and is becoming an option in the design of ships and in the analysis of new concepts. Nevertheless, it remains unclear to what extent ship designers rely on numerical predictions.

Examples of ship design process involving the use of advanced design and analysis methods (CAD/CFD) are reported in Bertram (2004). Besides giving examples of Simulation Based Design (SBD) environments, the paper review some less common applications of numerical simulations in ship design like aerodynamic analysis for the exhausts, fire simulations, ship evacuation simulation through Discrete Event Simulation (DES) techniques.

Stern et al. (2003) presents an overview of possible use of complex RANS equation solvers in advanced shape design, highlighting also the limits of present situation and indicating the lack of consensus on Verification and Validation of CFD and the lack of trained users, as slow-down factors in the diffusion of SBD.

Examples of RANS investigations used in the hydrodynamic design of ships are increasing by the hour and a complete list would be far beyond the dimension and scope of this report. Choi et al. (2004) used a RANS code to analyse the flow around a full appended Ro-Ro ferry design and decreasing the total resistance of the initial design, while Marzi and Gatchell (2003) used a RANS solver to guide the design of a fast RoPAX ferry. In Gorski and Coleman (2002), several advanced submarine sails of canopy-like shape are evaluated using a RANS code and improvements are analysed on the basis of the predicted secondary flow downstream of the drag.

9.3 Computer Integrated Design and Optimisation

The design process is greatly assisted by commercial CAD systems and most of these offer an easy support for grid generation too. Indeed, most of the grid generation system now in use read the CAD data via IGES files and convert all the surface patches into NURBS patches so that it is easy to check for gaps and/or overlaps. These grid generators are then able to produce a volume grid from these data. A new trend is also to integrate into the CAD software some programming tools so that the user may develop specific procedures tailored on his specific needs (Tronstad and Rodigruez, 2004).

CAD systems are also adopted during concept exploration. When this search phase begins (whether coupled or not with an optimisation process), the ship surface is parameterised using global and/or local variables, and a parametric hull description is obtained. Many CAD systems offer a (more or less simple) parametric description of the hull. Bertram (2004) report results from a benchmark study on a Ro-Ro ship among three CAD system largely diffused.

The Free Form Deformation technique (Sederberg and Parry, 1986) instead moves the hull surface directly. Initially developed for computer graphics applications, it has proved to be a robust, very flexible and easily implemented method in design codes.

9.4 Numerical Shape Optimisation

An essential sequel to simulation is its application in design, and the use of CFD-based shape optimisation in the naval context is definitively increasing. However, when compared to the diffusion and frequency of use in other industrial fields (aerospace, aeronautical, automotive), the number of its applications in ship design is still incomparably smaller. The Design engineers community seems to be sceptical about the usefulness of numerical optimisation. The simplicity of the optimisation problem solved when compared to the complexity of real-life design problems and the lack of experimental evidence of the success of the optimisation are still weak points. Nonetheless, navy designs more often involve the use of numerical shape optimisation than commercial designs, especially for complex redesign of unsatisfactory shapes.

From the standpoint of researchers and code developers, applications of numerical optimisation to ship design has evolved along two parallel path: those applicable in the early conceptual stages of the design, which adopt simple evaluation tools, and those focused on the detailed redesign of starting shape, which involve the use of CFD solvers.

An example of the first group is the paper by Lowe and Steel (2003), where genetic algorithms are used to explore the design space distinguishing between primary (length, beam, draft, displacement) and secondary (centres of flotation and buoyancy, prismatic coefficient, etc.) parameters. Grigoropoulos (2004) focused on deriving design trends for global parameters with respect to seakeeping performances, moving the local modification for improving resistance to a second phase of the design.

Optimisations papers relying on high-fidelity CFD solvers for detailed re-design present considerable more software challenges. One is the management of real-life constraints inside optimisation codes (Abt et al., 2004 and Campana et al., 2004).

Experimental confirmation of the success of the optimisation is now more often carried out. The optimised forms are tested in the towing tank and data are compared against the original hull form (Marzi and Gatchell, 2003, Campana et al., 2004 and Min et al., 2004). Tahara et al. (2004) solved the same problem of redesigning the bow region and the sonar dome of the same surface combatant.

Grid and geometry manipulation is another key point of shape optimisation. CAD-based approaches represent a frequent choice and Free-Form Deformation represent an attracting alternative, Tahara (2005), Jacquin et al. (2004), while other authors e.g. Brizzolara (2004) prefer the parametric representation of the hull surface.

Variable-fidelity models have been proposed to reduce the computational time. The alternate use of high- and low-fidelity prediction tools has a long history in industrial design and some recent papers adopt this approach (Yang and Löhner, 2004 and Campana et al., 2004).

Multi-objective algorithms, based on aggregated approaches or on the "Pareto" idea, are available and used by the code developers. Also global optimisation algorithms are adopted to prevent the optimisers to be trapped by local minima and non-convex feasible domains produced by nonlinear constraints, at the cost of increasing by one or two orders of magnitude the computational time (Minami and Hinatsu, 2002 and Peri and Campana, 2003). Metamodel approaches (also referred as surrogate models, such as neural networks and response surfaces) are available to reduce the overall computational time. Duvigneau and Visonneau (2004) adopted a combination of genetic algorithms and neural networks, performing a sensitivity study on the effects of turbulence models on the shape optimisation.

Looking at design optimisation in other industries, it can easily be recognised that Multidisciplinary Design Optimisation (MDO) is very widespread. MDO is a methodology for the design of systems where the interaction between several disciplines is fundamental and where the designer is free to significantly affect the system performance in more than one discipline.

The references collected here below aim at grouping some introductory papers about MDO. Modern methods in multi-disciplinary analysis in other industries are discussed in Alexandrov and Hussaini, 1997, Haslinger and Makinen, 2003, Kodiyalam and Sobieszczanski-Sobieski, 2001, Simpson et al., 2002 and Zang and Green, 1999.

Despite the number of papers and the growing recognition of its importance, much less effort has been put into ship design, even if this is basically a multidisciplinary problem.

9.5 Conclusions

The development of new ship concepts seems to be mainly driven by the desire for increased safety and speed, especially for passengers ships.

If compared to other engineering applications, the use of CFD in design methods of marine vehicles is far from being systematic, even if some industrial applications are starting to appear in the literature.

CFD is mostly used in a sequential way, testing the new designs and identifying unsuccessful shapes, therefore reducing the experimental effort. Eventually, the constant improvement of computing platforms and the diffusion of clusters for parallel computing will promote the embedding of RANS solvers in optimisation codes for shape optimisation.

Today, numerical shape optimisation is still at his infancy in ship hydrodynamics and ship designers are still very sceptical about its usefulness. It can be only registered that there is a constant growth of papers dealing with this topic together with some real industrial applications.

However, from the experience accumulated in other industries it is clear that multi-objective global optimisation and multidisciplinary design optimisation (MDO), will realise tangible improvements of the ship design process in the near future.

10. RECOMMENDATIONS FOR SCALING AND EXTRAPOLATION

10.1 Introduction

The speed-power prediction is the most important function of towing-tank facilities. ITTC's recommended procedure for power estimation is based on the full-scale resistance, whose measurement is very difficult and for which data is seldom available. Instead of full-scale resistance measurements, extrapolation of model-scale resistance to full-scale resistance is normally in use.

The 23rd ITTC RC conducted uncertainty analysis for extrapolation equations by treating the extrapolation equations as data-reduction equations and concluded that the uncertainty of the form factor has a key role in the propagation of error through the extrapolation of resistance from model to full-scale resistance. The present section focuses on uncertainty analysis for the form factor.

As a preliminary, Section 10.2 reviews uncertainty analysis of full-scale resistance. Section 10.3 presents the uncertainty associate with the determination of form factor by using a regression method. Section 10.4 presents a sample calculation of uncertainty analysis of form factor. Lastly, Section 10.5 provides conclusions.

10.2 Uncertainty Analysis of Full-Scale Resistance

Uncertainty of model-scale resistance will propagate into full-scale resistance through the extrapolation process. Uncertainty of full-scale resistance is analysed as follows.

Data Reduction Equations. The full-scale total resistance coefficient is given as following where air resistance and roughness allowance are excluded because they contain no uncertainty.

$$C_{TS} = (1+k)C_{FS} + C_R \tag{10.1}$$

$$C_R = C_{TM} - (1+k)C_{FM} \tag{10.2}$$

$$C_{TS} = (1+k)(C_{FS} - C_{FM}) + C_{TM} \tag{10.3}$$

Uncertainty Analysis for Total Resistance. The uncertainty in the total resistance coefficient can be expressed as follows.

$$U_{CTS}^2 = \left(\frac{\partial C_{TS}}{\partial k}U_k\right)^2 + \left(\frac{\partial C_{TS}}{\partial C_{FS}}U_{CFS}\right)^2 + \left(\frac{\partial C_{TS}}{\partial C_{FM}}U_{CFM}\right)^2 + \left(\frac{\partial C_{TS}}{\partial C_{TM}}U_{CTM}\right)^2 \tag{10.4}$$

$$U_{CTS}^2 = \left[(C_{FS} - C_{FM})U_k\right]^2 + \left[(1+k)U_{CFS}\right]^2 + \left[(1+k)U_{CFM}\right]^2 + \left[U_{CTM}\right]^2 \tag{10.5}$$

Thus we get the following equation.

$$U_{CTS}^2 = \left[(C_{FS} - C_{FM})U_k\right]^2 + \left[U_{CTM}\right]^2 \tag{10.6}$$

This equation shows that the uncertainty of the full-scale resistance consists of the uncertainty of the model-scale resistance and that of the form factor which is not measured but obtained from analysis.

10.3 Uncertainty Analysis for Form Factor

The form factor is given as follows.

$$C_{TM} = (1+k)C_{FM} + C_R \tag{10.7}$$

The usual uncertainty analysis cannot be applied to this equation because the two terms of the right side depend on each other.

A regression method is used to determine the magnitude of form factors. The uncertainty in form factors is associated with this process. When experimental information is represented by a regression, there are two kinds of uncertainties. One is the uncertainty due to the uncertainty in the original experimental results, the other is introduced if the wrong regression model is used. For the latter, discussion is not made here because ITTC recommended Prohaska's method for experimental evaluation of the form factor.

In the low speed region (say 0.1 < Fn < 0.2), the wave resistance component is assumed to be a function of F_n^4. The straight line plot of C_{TM}/C_{FM} versus F_n^4/C_{FM} will intersect the ordinate (F_n=0) at (1+k), enabling the form factor to be determined.

Prohaska gives the following expression,

$$\frac{C_T}{C_F} = m \cdot \frac{F_r^4}{C_F} + (1+k) \qquad (10.8)$$

Although the Eq. 10.8 is treated as a data reduction equation, special consideration should be introduced in the process of uncertainty analysis. Namely only the uncertainty due to the regression method is to be analysed. Usually, the least-squares estimation is introduced to determine the slope and intercept.

We assume the general linear equation as,

$$Y = mX + c \qquad (10.9)$$

Summation of deviations is given as follows:

$$\eta = \sum_{i=1}^{N} \left(Y_i - m \cdot X_i - c\right)^2 \qquad (10.10)$$

For η to be minimum,

$$\frac{\partial \eta}{\partial m} = 0 \text{ and } \frac{\partial \eta}{\partial c} = 0 \qquad (10.11)$$

Then we get,

$$c = \frac{\sum_{i=1}^{N}\left(X_i^2\right)\sum_{i=1}^{N} Y_i - \sum_{i=1}^{N} X_i \sum_{i=1}^{N}\left(X_i Y_i\right)}{N\sum_{i=1}^{N}\left(X_i^2\right) - \left(\sum_{i=1}^{N} X_i\right)^2} \qquad (10.12)$$

As mentioned at first, we deal not with the measured value but with the derived value. So it is better to introduce the classical linear regression uncertainty instead of the comprehensive regression uncertainty.

The standard error of regression is,

$$S_Y = \left[\frac{\sum_{i=1}^{N}\left(Y_i - mX_i - c\right)^2}{N-2}\right]^{1/2} \qquad (10.13)$$

$$\overline{X} = \frac{1}{N}\sum_{i=1}^{N} X_i \qquad (10.14)$$

$$S_{XX} = \sum_{i=1}^{N} X_i^2 - \frac{\left(\sum_{i=1}^{N} X_i\right)^2}{N} \qquad (10.15)$$

and that for the intercept is

$$S_C = \left[S_Y^2\left(\frac{1}{N} + \frac{\overline{X}^2}{S_{XX}}\right)\right]^{1/2} \qquad (10.16)$$

then we get,

$$C - 2S_C \le \mu_C \le C + 2S_C \qquad (10.17)$$

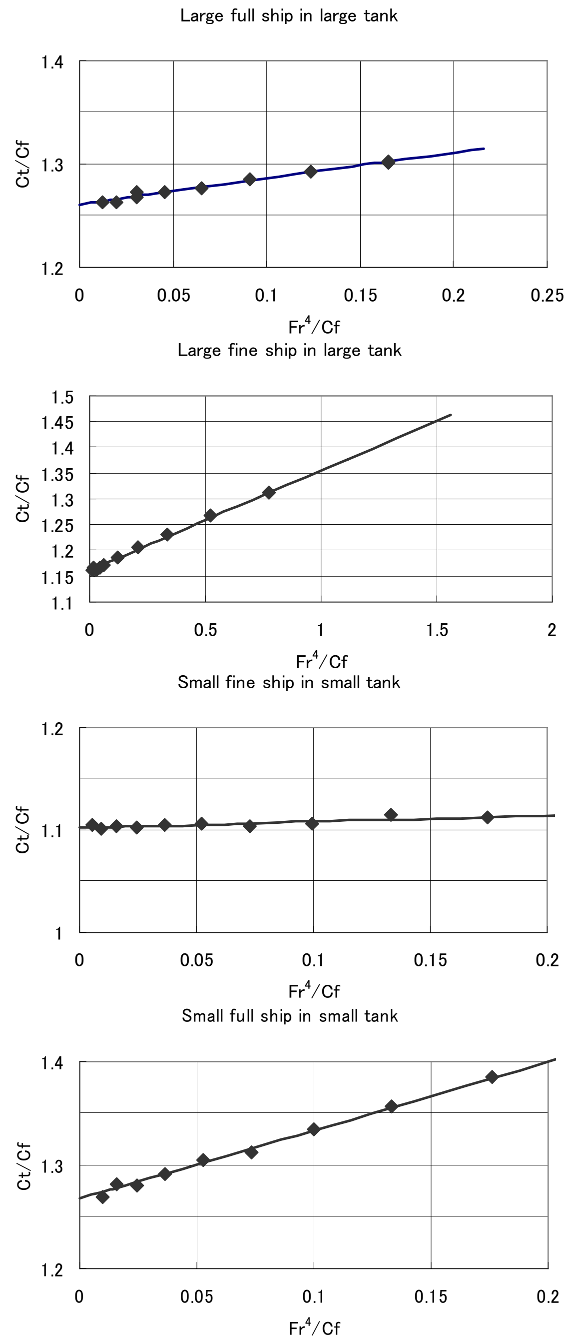
where,
C=1+k.

10.4 Sample Calculation of the Uncertainty of Full-Scale Resistance

For example, 4 kinds of resistance test results are analyzed by Prohaska's method and shown in Fig. 10.1.

In these examples, uncertainty in form factor is equal to 1% of k or 0.2% of 1+k.

From the Eq. 10.3 or 10.6, the magnitude of uncertainty due to the form factor is estimated to be 0.1% of C_{TS}.



		1+k	2Sc	% of 1+k
Large ship in large tank	Full ship L=6.8m ,Cb=.78	1.2611	0.0023	0.18
	Fine ship L=7.8m ,Cb=.57	1.1608	0.0028	0.24
Small ship in small tank	Full ship L=3.8m ,Cb=.76	1.2687	0.0032	0.25
	Fine ship L=4.0m ,Cb=.60	1.1021	0.0024	0.22

Figure 10.1- Examples of Prohaska's method.

10.5 Conclusions

The 23rd ITTC Resistance Committee showed that the two main causes of the uncertainty on the estimated full-scale resistance are the measured total resistance of the model ship and the analysed form factor. Since the uncertainty analysis on the total resistance of the model ship has already been conducted, the current RC carried out the uncertainty analysis on the analysed form factor. The uncertainty of the analysed form factor is due to the regression process of Prohaska's method, and the magnitude of uncertainty is nearly 0.1% of the total resistance.

11. RECOMMENDATIONS

11.1 Recommendations to the Conference

The Resistance Committee of the 24th ITTC has no Recommendations to the Conference.

12. REFERENCES AND NOMENCLATURE

12.1 References

Abt, C., Harries, S. and Hochkirch, K., 2004, "Constraint management for marine design application", PRADS 2004, Germany.

Aelbrecht, D., Dern, J.-C. and Doutreleau, Y. 2000, "A Design Tool for High Speed Ferries Wash", 23rd Symposium on Naval Hydrodynamics, Val de Reuil, France, pp. 957-967.

Alexandrov, N.M. and Hussaini, M.Y. (Eds.), 1997, "Multi-disciplinary Design Optimization - state of the art", Proc. of the ICASE / NASA Langley Workshop on Multidisciplinary Design Optimisation, SIAM Proc. Series.

Allenström, B., Andreasson, H., Leer-Andersen, M. and Li, D.-Q., 2003, "Amplification of ship generated wake wash due to coastal effects", Trans. SNAME, Vol. 111, pp. 61-77.

AIAA, 1995, Assessment of Wind Tunnel Data Uncertainty (S-071-1995), AIAA Standard Series.

AIAA, 1999, Assessment of Experimental Uncertainty with Application to Wind Tunnel Testing (S-071A-1999), AIAA Standard Series.

AIAA, 2003, AIAA Guide to Assessing Experimental Uncertainty-Supplement to S-071A-1999 (G-045-2003e).

Alin, N., Svennberg, U. and Fureby, C., 2003, "LES of the Flow Past Simplified Submarine Hulls", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Andrewartha, M., Doctors, L., Kantimanhanthi, K. and Brandner, P., 2003, "A numerical method for performance prediction of hydro-foil-assisted catamarans", FAST 2003, Italy.

Armstrong, T. and Holden, K., 2003, "A New Generation of Large Fast Ferry – From Concept to Contract Reality", FAST 2003, Italy.

ASME, 1998, Test Uncertainty, ASME Standard PTC 19.1-1998.

Azcueta, R., 2003, "Steady and unsteady RANSE simulations for planning crafts," Proc. 6th Numerical Towing Tank Symp., Rome, Italy.

Azcueta, R., 2004, "Steady and unsteady RANSE simulations for littoral combat ships," Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Babinsky, H., Kuschel, U., Hodson, H.P., Moore, D.F. and Welland, M.E, 2001, "Geometries for Five-Hole Type Probes with Planar Sensor Arrays", AIAA Journal, Vol.39, No.12, pp. 2414-2416.

Barone, M. and Bertorello, C., 2004, "Design Optimization of Trimaran Hull by Multiattribute Concept", PRADS 2004, pp. 270-277.

Bell, J.H., Schairer, E.T., Hand, L.A., and Mehta, R.D., 2001, Surface pressure measurements using luminescent coatings, Annual review of Fluid Mechanics, Vol.33, pp. 155-206.

Bensow, R., Persson, T., Fureby, C., Svennberg, U. and Alin, N., 2004, "Large eddy simulation of the viscous flow around submarine hulls," Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Bertram, V., 2004, "Toward Intelligent Simulation Based Design", COMPIT 2004, Siguenza (Spain).

Bertram, V. and Seif, M.S., 2004, "New developments for Fast and Unconventional Marine Vessels', HIPER 2004, Italy

Bourgoyne, D., Hamel, J.M., Ceccio, S.L. and Dowling, D.R., 2003, "Time-Averaged flow over a Hydrofoil at high Reynolds number", J. Fluid Mech., Vol. 496, pp. 365-404.

Brizzolara, S., 2004, "Parametric Optimization of SWAT-Hull Forms by a Viscous-Inviscid Free Surface Method Driven by a Differential Evolution Algorithm", 25th Symp. on Naval Hydro., St. John's, Canada.

Brizzolara, S., and Bruzzone, D., 2003, "Near and Distant Waves of Fast Ships in Unlimited and Limited Depths", FAST 2003, Ischia, Italy, Vol. 3, pp. H1.1-H1.12.

Brizzolara, S., Capasso, M. and Francescutto, A., 2003, "Effect of hulls form variations on the hydrodynamic performances in waves of a trimaran ship”, FAST 2003, Italy.

Broglia, R., Di Mascio, A. and Muscari, R., 2004, “Numerical simulations of breaking wave around a wedge”, Proc. 25th Symp. Naval Hydrodynamics, St. John’s, NL, Canada.

Brown, O.C., 2000, “Low-speed pressure measurements using a luminescent coating system”, PhD Thesis, Stanford Univ., Stanford.

Bucknall, R.W.G. and Greig, A.R., 2002, “The future of simulation in marine engineering design”, J. of Marine Design and Operations, Part B, No. B1, pp. 3-10.

Bull, P., Verkuyl, J.B., Ranocchia, D., Di Mas-cio, A., Dattola, R., Merle, L. and Cordier, S., 2002, “Prediction of high Reynolds number flow around naval vessels”, Proc. 24th Symp. Naval Hydrodynamics, Fukuoka, Japan.

Bulten, N.W.H. and Verbeek, R., 2003, “Design of optimal inlet duct geometry based on vessel operational profile”, FAST 2003, Italy.

Burg, C.O.E., Sreenivas, K., Hyams, D.G., and Mitchell, B., 2002, “Unstructured Nonlinear Free Surface Simulations for the Fully-Appended DTMB Model 5415 Series Hull Including Rotating Propulsors”, Proc. 24th Symp. Naval Hydrodynamics, Fukuoka, Japan, July.

Buttner, L. and Czarske, J., 2003, “Mutimode laser Doppler anemometer for turbulence measurements with high spatial resolution”, Proc. 4th ASME/JSME Joint Fluids Engineering Conference, Honolulu, Hawaii, USA.

Cadafalch, J., Perez-Segarra, C.D., Consul, R. and Oliva, A., 2002, “Verification of finite vol-ume computations on steady-state fluid flow and heat transfer”, J. Fluids Engineering, Vol. 124, pp. 11-21.

Calcagno, G., Di Felie, F., Felli, M. and Pereira, F., 2002, “Propeller Wake Analysis Behind a Ship by Stereo PIV”, Proc. of 24th Symposium on Naval Hydrodynamics, CD.

Campana, E.F., Peri, D., Tahara, Y. and Stern, F., 2004, “Comparison and Validation of CFD Based Local Optimization Methods for Surface Combatant Bow”, 25th Symp. on Naval Hydro., St. John’s, Canada.

Castillo, L., Seo, J., Hangan, H. and Johansson, T.G., 2004, “Smooth and rough turbulent boundary layers at high Reynolds number”, Experiments in Fluids Vol.36, No. 5, pp. 759-774.

Castillo, L., Seo, J., Hangan, H. and Johansson, T.G., 2004, “Smooth and Rough Turbulent Boundary Layers at High Reynolds Number,” Experiments in Fluids, No. 36, pp. 759-774.

Cavallo, P.A., Sinha, N. and Feldman, G.M., 2004, “Parallel Unstructured Mesh Adaptation for Transient Moving Body and Aeropropulsive Applications”, AIAA Paper No. 2004-1057.

Celik, I., Hu, G. and Badeau, A., 2003a, “Further refinement and benchmarking of a single-grid error estimation technique,” Paper No. AIAA-2003-0628.

Celik, I.B., Cehreli, Z.N., and Yavuz, I., 2003b, “Index of quality for large eddy simulations”, Proc. ASME FEDSM’03, FEDSM2003-45448, Honolulu, Hawaii.

Celik, I., and Hu, G., 2004, “Single grid error estimation using error transport equation”, to appear, J. of Fluids Engineering.

Chan, W.M., Gomez, R.J., Rogers, S.E. and Buning, P.G., “Best Practices in Overset

Grid Generation", AIAA Paper No. 2002-3191, June 2002.

Chen, P-F. and Huang, C-H., 2002, "An Inverse Hull Design Problem in Optimizing the Desired Wake of a Ship", J. Ship Research, Vol. 46, No. 2, June, pp. 138-147.

Chen, X.B., 2002, "Role of Surface Tension in Modelling Ship Waves", 17th International Workshop on Water Waves and Floating Bodies, Cambridge, England, August.

Choi, J-E., Min, K-S., Chung, S-H. and Seo, H-W., 2003, "Study on the scale effects on the flow characteristics around a full slow-speed ship", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Choi, S.H., Kim, J.J., Ahn, S.M., Jang, H.S. and Joo, Y.R., 2004, "Hydrodynamics design consideration of medium sise Ro-Ro passenger ferry", PRADS 2004, Germany.

Chun, H.-H., Park, J.-C., Choi, H.-J., Yoon, H.-S. and Kang, D.-H., 2004, "Numerical study on turbulent flow around ship models using a large-eddy simulation technique", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Colagrossi, A., Lugni, C., Greco, M. and Faltinsen, O.M., 2004, "Experimental and Numerical Investigation of 2D Sloshing with Slamming", 19th International Workshop on Water Waves and Floating Bodies, Cortona, Italy, March.

Coleman, H.W. and Steele, W.G., 1999, Experimentation and uncertainty analysis for engineers, 2nd Ed. John Wiley & Sons, Inc, New York.

Cowen, E.A., Chang, K.-A. and Liao, Q., 2001, "A Single-camera coupled PTV-LIF technique", Experiments in Fluids, Vol. 31, pp. 63-73.

Cristensen, K.T. and Adrian, R.J., 2002, "Measurement of instantaneous Eulerian acceleration fields by particle image accelerometry; method and accuracy", Experiments in Fluids, Vol. 33, pp. 759-769

Dailey, H. and Ebert, M.P., 2003, "Comparison of Unstructured and Structured RANS Computations for David Taylor Model Body-1", NSWCCD-50-2003/39.

Day, A., Clelland, D. and Nixon, E., 2003, "Experimental and numerical investigations of 'Arrow' trimarans", FAST 2003, Italy.

Day, A.H. and Doctors, L.J., 2001, "Rapid Estimation of Near- and Far-field Wave Wake from Ships and Application to Hull-form Design and Optimization", J. Ship Research Vol. 45, Number 1, pp. 73-84.

DeGraaff, D.B. and Eaton, J.K., 2001, "A highresolution laser Doppler anemometer: design, qualification and uncertainty", Experiments in Fluids, Vol. 32, pp. 522-530.

Di Masco, A., Muscari, R. and Broglia, R., 2003, "Computation of free surface flows around ship hulls by a level-set approach", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Dietz, W., Wang, L., Wenren, Y., Caradonna, F. and Steinhoff, J., 2004, "The Development of a CFD-based model of Dynamic Stall", Annual Forum of the American Helicopter Society, Baltimore, June.

Doctors, L.J. and Day, A.H., 2000, "Wave-Free River-Based Air-Cushion Vehicles", Proc. RINA Intl. Conference on Hydrodynamics of High Speed Craft Wake Wash and Motions Control, London.

Doctors, L.J. and Day, A.H., 2001, "The Generation and Decay of Waves behind High-Speed Vessels", 16th IWWWFB, Hiroshima, Japan, pp. 33-36.

Doctors, L.J. 2003, "The influence of Viscosity on the Wavemaking of a Model Catamaran", 18th IWWWFB, Le Croisic, France.

Doctors, L.J. and Scrace, R.J., 2003, "The optimization of trimaran sidehull position for minimum resistance", FAST 2003, Ischia, Italy.

Doctors, L.J., 2004, "A Comparison of the Environmental Wave Generation of Hovercraft and other High-Speed Vessels", Proc. 5th Air-Cushion Technology Conference and Exhibition, Hovercraft Museum, Lee-on-Solent, UK.

Doctors, L.J. and Zilman, G., 2004a, "The influence of Surface Tension and Viscosity on the Wavemaking of a Model Catamaran", 19th IWWWFB, Cortona, Italy, pp. 11.1-11.4.

Doctors, L.J. and Zilman, G., 2004b, "Environmental Wave Generation of High-Speed Marine Vessels", Proc. 25th Symposium on Naval Hydrodynamics, St Johns, Canada.

Dommermuth, D.G., Sussman, M., Beck, R.F., O'Shea, T.T., Wyatt, D.C., Olson, K., and MacNeice, P., 2004, "The Numerical Simulation of Ship Waves Using Cartesian Grid Methods with Adaptive Mesh Refinement", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Doyle, R., Whittaker, T.J.T., and Elsaesser, B., 2001, "Wake wash characteristics of high-speed ferries in shallow water", Ship Technology Research, Vol. 48, pp. 181-196

Durbin, P.A., Medic, G., Seo, J.-M., Eaton, J.K. and Song, S., 2001, "Rough Wall Modifcation of Two-Layer k-ε", Journal of Fluids Engineering, 123:16-21.

Duvigneau, R. and Visonneau, M., 2004, "Hybrid genetic algorithms and artificial neural networks for complex design optimisation in CFD", Int. J. for Num. Methods in Fluids, Vol. 44, 1257-1278.

Duvigneau, R., Visonneau, M. and Deng, G.B., 2003, "On the Role Played by Turbulence Closures in Hull Shape Optimization at Model and Full Scale", J. Mar Sci Technology, Vol. 8, pp. 11-25.

Ebert, M.P., Gorski, J.J. and Coleman, R.M., 2003, "Viscous flow calculations with waterjet propelled ships", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Eefsen, T., van Walree, F., Peri D., van Terwisga, P., Kristensen, H.O., Dattola, R. and Visser, M., 2004, "Development of frigate designs with good seakeeping characteristics", PRADS 2004, Germany.

Eça, L. and Hoekstra, M., 2002, "An evaluation of verification procedures for CFD applications", Proc. 24th Symp. Naval Hydrodynamics, Fukuoka, Japan.

Faltinsen, O.M., Lugni, C. and Landrini, M., 2003, "Hydrodynamic Aspects of High-Speed Vessels", FAST 2003, Italy.

Fan, M. and Steinhoff, J., 2004, "Computation of Blunt Body Wake Flow by Vorticity Confinement", AIAA Paper No. 2004-0592, AIAA Conference, Reno, Nevada, January.

Fan, M., Wenren, Y., Dietz, W., Xiao, M. and Steinhoff, J., 2002, "Computing Blunt Body Flows on Coarse Grids Using Vorticity Confinement", ASME Journal of Fluids Engineering, Vol. 124, No. 4, December.

Floden, D., Keunjae, K. and Ottoson, P., 2004 "A Computational / Experimental Investigation on Resistance and Seakeeping Characteristics of Trimaran Configuration in Comparison with Monohull", PRADS 2004, 278-285, Germany.

Forliti, D.J., Strykowski, P.J. and Debatin, K., 2000, "Bias and precision errors of digital particle image velocimetry", Experiments in Fluids, Vol. 28, pp. 436-447.

Fu, T., Atsavaprance, P. and Hess, D., 2002, "PIV Measurements of the Cross-Flow Wake of a Turning Submarine Model (ONR Body 1)", Proceedings of 24th Symposium on Naval Hydrodynamics, CD-ROM.

Furey, D. and Fu, D., 2002, "Quantitative Visualization (QViz) Hydrodynamic measurement Techniques of Multiple Unsteady Surfaces", Proceedings of 24th Symposium on Naval Hydrodynamics, CR-ROM.

Gorski, J.J., 2002a, "Present State of Numerical Ship Hydrodynamics and Validation Experiments", Journal of Offshore Mechanics and Arctic Engineering, Vol. 124, pp. 74-80.

Gorski, J.J., 2002b, "A Perspective on the Role of RANS Codes for Predicting Large Amplitude Ship Motions", Proc. 6th Int. Ship Stability Workshop, Glen Cove, New York.

Gorski, J.J. and Coleman, R.M., 2002, "Use of RANS Calculations in the Design of a Submarine Sail", NATO RTO Symp. on Reduction of Military Vehicle Acquisition Time and Cost Through Advanced Modelling and Virtual Product Simulation, Paris, France.

Gorski, J.J., Haussling, H.J., Percival, A.S., Shaughnessy, J.J. and Buley, G.M., 2002, "The use of a RANS code in the design and analysis of a naval combatant", Proc. 24th Symp. Naval Hydrodynamics, Fukuoka, Japan.

Gorski, J.J., Miller, R.W. and Coleman, R.M., 2004, "An investigation of propeller inflow for naval surface combatants", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Grigoropoulos G.J., 2004, "Hull form optimisation for hydrodynamic performances", JSR, Vol. 41, No. 4, 167-182.

Gui, L., Longo, J., Metcalf, B. , Shao, J.H. and Stern, F., 2001, "Forces, Moment, and Wave Pattern for surface combatant in Regular Head Waves Part I. Measurement Systems and Uncertainty Assessment", Experiments in Fluids Vol.31, No, 6 pp. 674-680.

Gui, L., Longo, J. and Stern, F., 2001, "Towing tank PIV measurement system, data and uncertainty assessment for DTMB Model 5512", Experiments in Fluids, Vol. 31, pp. 336-346.

Gui, L., Longo, J., Metcalf, B., Shao, J.H. and Stern, F., 2002, "Forces, Moment, and Wave Pattern for surface combatant in Regular Head Waves. Part II .Measurement Results and Discussions", Experiments in Fluids Vol. 32, No, 1 pp. 27-36.

Guillerm, P-E. and Alessandrini, B., 2003, "3D Free-Surface Flow Computation Using a RANSE/Fourier-Kochin Coupling", Int. J. Numer. Meth. Fluids, Vol. 43, pp. 301-318.

Hanel, D., Stuntz, N., Steffen, R., Jiang, T., Henn, R., Stoye, T. and Vilsmeier, R., 2003, "Close Coupling of Viscous 3D Flow and Shallow Water Wave Simulations", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Haslinger, J. and Makinen, R.A.E., 2003, "Introduction to Shape Optimization - Theory, Approximation and Computation", SIAM, Advances in Design and Control series.

Hay, A., Queutey, P. and Visonneau, M., 2004, "Computation of three-dimensional free-surface flows with an automatic mesh refinement and coarsening strategy", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Hemsch, M.J. and Tuttle, D.G., 2004, "Measurement of force balance repeatability and reproducibility in the NTF", AIAA 2004-771.

Hino, T., Ohashi, K. and Ukon, Y., 2003, "Flow computations around a ship with appendages by an unstructured grid based NS solver", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Hino, T., 2004, "Numerical simulations of breaking waves around an advancing ship", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Hong, C-B., Doi, Y. and Mutsuda, H., 2003, "Numerical study on ship wave breaking in restricted waterway", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Hu, C., Kashiwagi, M. and Kishev, Z., 2004, "Numerical Simulation of Violent Sloshing by CIP Method", 19th IWWWFB, Cortona, Italy, March.

Iafrati, A. and Campana, E., 2004, "Spectral Analysis of the Free Surface Fluctuations behind Microbreakers", 19th IWWWFB, 28-31 March, Cortona, Italy.

Ikemoto, A., Kato, H. and Yoshida, Y., 2002, "Development of Void Fraction Measuring Device for Full-scale Test", J. of Soc. Nav. Arch. of Japan, Vol.191, pp. 43-50 (in Japanese).

Imas, L., Grant, J., Kring, D. and Milewski, D. M., 2003, "Investigation of free-surface flow in the near-field of an advancing surface-piercing body using three- dimensional smoothed particle hydrodynamics", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Imas, L., 2004, "Evaluation of the SPH method for hydrodynamic applications; single and multiphase flows", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Jacquin, E., Derbanne, Q., Bellevre, D., Cordier, S., Alessandrini, B. and Roux, Y., 2004, "Hull form optimization using a free surface RANSE solver", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Janson, C.-E., Leer-Andersen, M., and Larsson, L., 2003 "Calculation of Deep-Water Wash Waves Using a Combined Rankine/Kelvin Source Method", J. Ship Research, Vol. 47, No. 4, pp. 313-326.

Jiang, T., 2001, "Ship Waves in Shallow Water", Frotschritt-Berichte VDI, Series 12, No. 466.

Jiang, T., Henn, R. and Sharma, S.D., 2002, "Wash Waves Generated by Ships Moving on Fairways of Varying Topography", 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.

Jimènez, J., 2004, "Turbulent Flows over Rough Walls", Annual Review Fluid Mechanics, No. 36:173-96.

Kieft, R.N., Schreel, K.R.A.M., Van der Plas, G.A.J. and Rindt, C.C.M., 2002, "The application of a 3D PTV algorithm to a mixed convection flow", Experiments in Fluids, Vol. 33, pp. 603-611.

Kim, W.J., Van, S.H. and Kim, D.H., 2001, "Measurement of Flows around Modern Commercial Ship Models", Experiments in Fluids Vol.31, No, 5 pp. 567-578.

Kodama, Y., Kakugawa, A., Takahashi, T., Ishikawa, S., Kawakita, C., Kanai, T., Toda, Y., Kato, H., Ikemoto, A., Yamashita, K. and Nagamatsu, T., 2002, "A Full-scale Experiment on Microbubbles for Skin Friction Reduction Using "SEIUN MARU" Part1: The Preparatory Study", J. of Soc.

Nav. Arch. Japan, Vol.192, pp. 1-14 (in Japanese).

Kodama, Y., Kakugawa, A., Takahashi, T., Nagaya, S., Sugiyama, K. and SR239 Committee of the Shipbuilding Research Association of Japan, 2002, "Microbubbles: Drag Reduction Mechanism and Applicability to Ships", 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan, July.

Kodiyalam, S. and Sobieszczanski-Sobieski, J., 2001, "Multidisciplinary Design Optimization - some formal methods, Framework Requirements, and Application to Vehicle Design", Int. J. Vehicle Design (Special Issue), pp. 3-22.

Kofoed-Hansen, H., Jensen, T., Kirkegaard, J. and Fuchs, J., 1999, "Prediction of wake wash from high-speed craft in coastal areas", RINA Intl. Conference on Hydrodynamics of High Speed Craft, London.

Kotey, N.A., Bergstrom, D.J. and Tachie, M.F., 2003, "Power Laws for Rough Wall Turbulent Boundary Layers", Physics of Fluids, Vol. 15, No. 6.

Kume, K., Hasegawa, J. and Tukada, Y., 2005, "Measurements of hydrodynamic forces, surface pressure and wake for obliquely towed KVLCC2M model and uncertainty analysis", CFD Workshop Tokyo 2005.

Kyoung, J., Bai, K.J. and Kim, J.W., 2003, "Nonlinear Free-Surface Flows around Three Dimensional Transom Stern", The 8th International Conference on Numerical Ship Hydrodynamics, Pusan, Korea.

Lang, N. and Limberg, W., 1999, "Construction of three-dimensional flow structure out of two-dimensional steady flow field velocity measurement", Experiments in Fluids, vol. 27, pp. 351-358.

Langtry R.B., Menter F.R., Likki, S.R., Suzen Y.B., and Huang, P.G., 2004, "A Correlation-Based Transition model using Local Variables Part II – Test Cases and Industrial Application", Proc. of the ASME Turbo Expo 2004, Vienna, Austria, June.

Langtry, R.B. and Menter, F.R., 2005, "Transition Modeling for General CFD Applications in Aeronautics", AIAA Paper 2005-522 Reno, Nevada.

Larsson, L., Stern, F., and Bertram, V., 2003, "Benchmarking of Computational Fluid Dynamics for Ship Flows: The Gothenburg 2000 Workshop", J. Ship Research, Vol. 47, No. 1, March, pp. 63-81.

Launder, B.E. and Sandham, N.D., (Eds), 2002, Closure Strategies for Turbulent and Transitional Flows, Cambridge University Press, ISBN-10 0521792088 ISBN-13 9780521792080.

Lavoie, P. and Pollard, A., 2003, "Uncertainty analysis of four-sensor hot-wires and their data-reduction schemes used in the near field of a turbulent jet", Experiments in Fluids, Vol. 34(3), pp. 358-370.

Lee, S., Paik, B. and Lee, C., 2002, "Phase-Averaged PTV Measurements of Propeller Wake", Proc. of 24th Symposium on Naval Hydrodynamics, CD.

Leer-Andersen, M. and Larsson, L., 2003, "An Experimental/Numerical Approach for Evaluating Skin Friction on Full-Scale Ships with Surface Roughness", J. Marine Science and Technology, Vol. 8, pp. 26-36.

Leer-Andersen, M. and Larsson, L., 2003, "An Experimental/Numerical Approach for Evaluating Skin Friction on Full-Scale Ships with Surface Roughness", Journal of Marine Science and Technology, 8:26-36.

Li, T., 2002, "Ship Wavemaking in the Context of a RaNS-Solver FINFLO-SHIP", Report M-270, Ship Laboratory, Helsinki University of Technology.

Li, T., 2003, "Computation of Turbulent Free-Surface Flows Around Modern Ships", Int. J. Numer. Meth. Fluids, Vol. 43, pp. 407-430.

Lillberg, E. and Svennberg, U., 2004, "Large eddy simulation of the viscous flow around a ship hull including the free-surface", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Löhner, R., Baum, J.D. and Mestreau, E., 2004, "Advances in Adaptive Embedded Unstructured Grid Methods," AIAA Paper No. 2004-83.

Löhner, R., Yang, C. and Roger, R., 2002, "Tracking vortices over large distances using vorticity confinement", Proc. 24th Symp. on Naval Hydrodynamics, Fukuoka, Japan.

Lowe, T.W. and Steel, J., 2003, "Conceptual hull design using a genetic algorithm", JSR, Vol. 47, No. 3, pp. 222-236.

Lubke, L.O. and Mach, K-P., 2004, "Angular based LDA measurements in the wake of the propelled KCS model and its use to validate CFD calculations", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Luo, H., Sharov, D., Baum, J.D. and Lohner, R., 2000, "On the Computation of Compressible Turbulent Flows on Unstructured Grids", AIAA Paper No. 2000-0927, Jan.

Macfarlane, G.J. and Renilson, M.R., 2000, "When is Low Wash Low Wash? – An Investigation Using a Wave Wake Database", RINA Intl. Conference on Hydrodynamics of High Speed Craft Wake Wash and Motions Control, London.

Marine Accident Investigation Branch (MAIB), 2004, "Report on the investigation of the swamping and foundering of a 4.6m grp open sports boat with the loss of three lives on Loch Ryan South-west Scotland 12 July 2003", Marine Accident Investigation Branch Report No 5/2004, Southampton UK.

Martin, P.B., Pugliese, G.J. and Leishman, J.G., 2000, "Laser Doppler velocimetry uncertainty analysis for rotor blade tip vortex measurements", Proc 38th Aerospace Science meeting/Applied Aerodynamics, AIAA 2000-0263.

Marzi, J. and Gatchell, S., 2003, "Hullform analysis and optimisation for a fast RoPAX ferry", FAST 2003, Italy.

McKenna, S.P. and McGillis, W.R., 2002, "Performance of digital image velocimetry processing technique", Experiments in Fluids, Vol. 33, pp. 759-769.

Menter, F.R, 1993, "Zonal Two-Equation k-ω Turbulence Models for Aerodynamic Flows", AIAA Paper 93-2906, AIAA Conference, Orlando, Florida, July.

Menter, F.R, Langtry, R.B., Likki, S.R., Suzen Y.B. and Huang, P.G., 2004, "A Correlation -Based Transition model using Local Variables Part I – Model Formulation", Proc. of the ASME Turbo Expo 2004, Vienna, Austria, June.

Mewis, F. and Klug, H., 2004, "The challenge of Very Large Container Ships – A Hydrodynamic View", PRADS 2004, pp. 173-181, Germany.

Meyn, L.A., 2000, "An uncertainty propagation methodology hat simplifies uncertainty analyses", AIAA-2000-0149.

Min, K.-S., Lee, Y.-S., Kang, S.-H. and Han, B.-W., 2004, "Theoretical Hull Optimization for Fine Higher-Speed Ships", 25th Symp. on Naval Hydro., St. John's, Canada.

Minami, Y. and Hinatsu, M., 2002, "Multi-Objective Optimization of Ship Hull

Form Design by Response Surface Methodology", 24th Symp. on Naval Hydro., Fukuoka, Japan.

Molland, A.F., Wilson, P.A., Taunton, D.J., Chandraprabha, S. and Ghani, P.A., 2004, "Resistance and Wash Wave Measurements on a Series of High Speed Displacement Monohull and Catamaran Forms in Shallow Water", Int. J. Maritime Engineering.

Muscari, R., and Di Mascio, A., 2003a, "Modeling of Breaking Waves around Ships", 6th Numerical Towing Tank Symposium, Rome, Italy.

Muscari, R., and Di Mascio, A., 2003b, "A Model for the simulation of Steady Spilling Breaking Waves", Journal of Ship Research, Vol. 47, No. 1, pp. 13-23.

Nagamatsu, T., Kodama, Y., Kakugawa, A., Takai, M., Murakami, K., Ishikawa, S., Kamiirisa, H., Ogiwara, F., Yoshida, Y., Suzuki, T., Toda, Y., Kato, H., Ikemoto, A., Yamatani, F., Imo, F. and Yamashita, K., 2002, "A Full-scale Experiment on Microbubbles for Skin Friction Reduction Using "SEIUN MARU" Part2: The Full-scale Experiment", J. of Soc. Nav. Arch. Japan, Vol.192, pp. 15-28 (in Japanese).

Nelson, C.C. and Roy, C.J., 2004, "Verification of the Wind-US CFD code using the method of manufactured solutions", AIAA Paper No. 2004-1104.

Nowak, M., 2002, "Wall Shear Stress Measurement in a Turbulent Pipe Flow using Ultrasound Doppler Velocimetry", Experiments in Fluids Vol.33, No, 2 pp. 249-255.

Olivieri, A., Pistani, F. and Di Mascio, A., 2003, "Breaking wave at the bow of a fast displacement ship model", J. of Mar. Sci. Techn., Vol. 8, pp. 68-75.

Olivieri, A., Pistani, F. and Penna, R., 2003, "Experimental investigation of the flow around a fast displacement ship hull model", Journal of Ship Research, Vol.47, No.3, pp. 247-261.

Park, I-R, Van, S-H. and Ahn, H-S., 2004, "Two-phase viscous flow simulation around a commercial container ship hull form", J. of Ships & Ocean Engineering, Vol. 37, pp. 147-154, April.

Patel, V.C., 1998, "Perspective: Flow at High Reynolds Numbers and over Rough Surfaces – Achiles Heel of CFD", Journal of Fluids Engineering, 120:434-44.

Payer, H.G., 2004, "Challenges in ship design, fabrication and inspection – a class view", PRADS 2004, Germany.

Pelletier, D., Turgeon, E., Lacasse, D. and Borggaard, J., 2003, "Adaptivity, sensitivity, and uncertainty: toward standards of good practice in computational fluid dynamics", AIAA Journal, Vol. 41, No. 10, pp. 1925-1933.

Pereira, F., Costa, T., Felli, M., Calcagno, G. and Felice, F., 2003, "A versatile fully submersible stereo-PIV probe for towing tank application", Proc. 4th ASME/JSME Joint Fluids Engineering, Conference, Honolulu, Hawaii, USA.

Peri D. and Campana E.F., 2003a, "High-fidelity Models in Simulation Based Design", 8th Int. Conf. on Numerical Ship Hydrodynamics, Busan, Korea.

Peri D. and Campana E.F., 2003b, "Multidisciplinary Design Optimization of a Naval Surface Combatant", J. Ship Research, Vol. 47, No.1, 1-12.

Petrovic, D.V., Vukoslavcevic, P.V. and Wallace, J.M., 2003, "The accuracy of turbulent velocity component measurements by multi-sensor hot-wire probes; a new approach to an old problem", Experiment in Fluids, Vol. 34, pp.130-139.

Proceedings, "Int. Conf. On Design and Operation of Trimaran Ships", 2004, London (UK).

Qin, Y. and Shih, T.I-P., 2002, "A discrete transport equation for error estimation in CFD", AIAA Paper No. 2002-0906.

Qin, Y., Keller, P.S., Sun, R.L., Hernandez, E. C., Perng, C.-Y., Trigui, N., Han, Z., Shen, F.Z., Shieh, T., and Shih, T.I-P., 2004, "Estimating grid-induced errors in CFD by discrete-error-transport equations", AIAA Paper No. 2004-656.

Raupauch, M.R., Antonia, R.A. and Rajagopalan, S., 1991, "Rough-Wall Turbulent Boundary Layers", Journal of Applied Mechanics Rev., No. 44, pp. 1-25.

Raven, H.C., 2000, "Numerical Wash Prediction using a Free-Surface Panel Code", Proc. RINA Intl. Conference on Hydrodynamics of High Speed Craft Wake Wash and Motions Control, London.

Raven, H.C. and Starke, B., 2002, "Efficient methods to compute steady ship viscous flow with free surface", Proc. 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.

Raven, H.C., van der Ploeg, A. and Starke, B., 2004, "Computation of free-surface viscous flows at model and full scale by a steady iterative approach", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Reis, M.L.C.C., Mello, O.A.F. and Uyeno, S., 2002, "A method for estimation of the calibration uncertainty of an external six-component wind tunnel balance", AIAA 2002-2793.

Reis, M.L.C.C., Mello, O.A.F. and Uyeno, S., 2003, "Calibration uncertainty of an external six-component wind tunnel balance", AIAA 2003-3884.

Renilson, M., Scrace, B., Johnson, M. and Richardsen, C., 2004, "Trials to measure the hydrodynamic performance of RV Triton", Int. Conf. On Design and Operation of Trimaran Ships, London (UK).

Rhee, S.H. and Stern, F., 2001, "Unsteady RANS Method For Surface Ship Boundary Layer And Wake And Wave Field", Int. J. Numerical Methods In Fluids, Vol. 37, pp. 445-478.

Roache, P., 1998, Verification and Validation in Computational Science and Engineering, Hermosa Publishers, Albuquerque, NM.

Roache, P.J., 2002, "Code verification by the method of manufactured solutions", J. of Fluids Engineering, Vol. 124, pp. 4-10.

Roache, P.J., 2003, "Criticisms of the "correction factor" verification method", J. of Fluids Engineering, Vol. 125, pp. 732-733.

Roache, P.J., 2004a, "Building PDE codes to be verifiable and validatable", Computing in Science and Engineering, Sept./Oct, pp. 30-38.

Roache, P.J., 2004b, "Overview of "Workshop on CFD Uncertainty Analysis", 21-22 October, 2004, Instituto Superior Técnico, Lisbon, Portugal", private communication.

The Royal Institution of Naval Architects, 2002, "Drag Reduction Efforts Coming to Fruition?", Warship Technology, International Journal of The Royal Institution of Naval Architects, October.

Ruan, X., Song, X. and Yamamoto, F., 2001, "Direct measurement of the vorticity field in digital particle images", Experiment in Fluids, Vol. 30, pp. 696-704.

Sahoo, P.K., Browne, N.A. and Salas, M., 2004, "Experimental and CFD study of wave resistance of high-speed round bilge catamaran hull forms", 4th Int. Conf. High-Performance Marine Vehicles, Rome, Italy.

Saisto, I. 2000, "Computation of Two-Dimensional Transom Stern Wave using the Boundary Element Method", Licentite's Thesis, Ship Laboratory, Helsinki University of Technology.

Schultz, M.P., 2002, "The Relationship between Frictional Resistance and Roughness for Surfaces Smoothed by Sanding", J. of Fluids Engineering ASME, Vol.124, No.2, pp. 492-499.

Schweighofer, J., 2003, "Investigation of Two-Dimensional Transom Waves Using Inviscid and Viscous Free-Surface Boundary Conditions at Model- and Full-Scale Ship Reynolds Numbers", Report M-281, Dissertation, Ship Laboratory, Helsinki Univ. of Technology.

Sederberg, T. and Parry, S., 1986, "Free-Form Deformation of Solid Geometric Models", Computer Graphics, 20, 4, pp. 151-160.

Sethian, J. A. and Smereka, P., 2003, "Level set methods for fluid interfaces", Annual Rev. Fluid Mechanics, Vol. 35, pp. 341-372.

Simonsen, C.D., 2000, "Rudder, Propeller And Hull Interaction By Rans", Ph.D. Thesis, Deaprtment Of Naval Architecture And Offshore Engineering, Danish Technical University, Lyngby, Denmark.

Simonsen, C.D. and Stern, F., 2003a, "Verification and validation of RANS maneuvering simulation of Esso Osaka: effects of drift and rudder angle forces and moments", Computers and Fluids, Vol. 32, pp. 1325-1356.

Simonsen, C.D. and Stern, F., 2003b, "Flow pattern around an appended tanker hull form in simple maneuvering conditions", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Simpson, T.W., Brooker, A.J., Ghosh, D., Giunta, A.A., Koch, P.N. and Yang, R.-J., (2002), "Approximation Methods in Multidisciplinary Analysis and Optimization: A Panel Discussion", document which summarises the discussion at 9th AIAA/ISSMO Symp. on Multidisciplinary Analysis & Optimization, 2-4/09/2002, Atlanta, Georgia, USA.

Steinhoff, J., Dietz, W., Haas, S., Xiao, M., Lynn, N. and Fan, M., 2003a, "Simulating Smal-Scale Features in Fluid Dynamics and Accoustics as Nonlinear Solitary Waves", AIAA Paper No. 2003-0078, AIAA Conference, Reno, Nevada.

Steinhoff, J., Fan, M., Wang, L. and Dietz, W., 2003b, "Convection of Concentrated Vortices and Passive Scalars as Solitary Waves", Journal of Scientific Computing, Vol. 19, Nos. 1-3.

Stern, F., Wilson, R.V., Coleman, H.W. and Paterson, E.G., 2001, "Comprehensive approach to verification and validation of CFD simulations – Part I: methodology and procedures", J. Fluids Engineering, Vol. 123, pp. 793-802.

Stern, F., Olivieri, A., Shao, J., Longo, J., and Ratcliffe, T., "Statistical Approach for Estimating Intervals of Certification or Biases of Facilities or Measurement Systems Including Uncertainties", to be published in ASME J. of Fluids Eng.

Stern F., Wilson, R., Longo, J., Carrica, P., Xing, T., Tahara, Y., Simonsen, C., Kim, J., Shao, J., Irvine, M., Kandysamy, M., Ghosh, S. and Weymouth, G., 2003, "Paradigm for development of simulation based design for ship hydrodynamics", 8th Int. Conf. on Num. Ship Hydrodynamics, Busan, Korea.

Subramanian, C.S., Shinjo, N. and Gangadharan, S.N., 2004, "A Study of Hydrodynamic Characteristics of Boundary Layer with Algae Roughness", Marine Technology, Vol. 41, No. 2, pp. 60-66.

Tachie, M.F., Bergstrom. D.J., Balachandar R. and Ramachandran, S., 2001, "Skin Friction Correlation in Open Channel Boundary Layers", J. of Fluids Engineering ASME, Vol.123, No.4, pp. 953-954.

Tahara, Y., Katsui, T. and Himeno, Y., 2002, "Computation of Ship Viscous Flow at Full Scale Reynolds Number", J. of the Soc. Naval Architects of Japan, Vol. 192, Dec., pp. 89-101.

Tahara, Y., Peri, D., Campana, E.F. and Stern, F. 2005, "CFD-Based Multiobjective Optimization of a Surface Combatant", Osaka Colloquium, Japan.

Tahara, Y., Stern, F. and Himeno, Y., 2004, "Computational fluid dynamics-based optimization of a surface combatant", J. Ship Research, Vol. 28, No. 4, pp. 273-287.

Takada, N., Hoshino, T., Ishikawa, S. and Higaki, S., 2002, "Simulation of Free-Surface Viscous Flow Around Practical Hull and Rudder with Propeller Effects", J. of the Soc. Naval Architects of Japan, Vol. 192, Dec., pp. 63-70.

Tanaka, H., Toda, Y., Higo, K. and Yamashita, K., 2003, "Influence of Surface Properties of Coatings to Frictional Resistance", J. Kansai Soc. N.A., Japan, No.239, pp. 21-28

Tronstad, R. and Rodriguez, A., 2004, "Automatic tools in the design process", COMPIT 2004, Spain.

Tuck, E.O., Scullen, D.C. and Lazauskas, L., 2000, "Ship-Wave Patterns in the spirit of Michell", Proc IUTAM Symposium on Free-Surface Flows, Birmingham, UK.

Tuck, E.O., Scullen, D.C. and Lazauskas, L., 2002, "Wave Patterns and Minimum Wave Resistance for High-Speed Vessels", 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.

Upatnieks, A., Laberteaux, K. and Ceccio, S. L., 2002, "A kilohertz frame rate cinemagraphic PIV system for laboratory-scale turbulent and unsteady flows", Experiments in Fluids, Vol. 32, pp. 87-98.

Valsgard, S., Reepmayer, O., Lothe, P., Strom, N.K. and Mork, K., 2004, "The development of a Compressed Natural Gas Carrier", PRADS, pp. 163-172, Germany.

Van, S-H., Kim, J., Park, I-R. and Kim, W-J., 2003, "Calculation of turbulent flows around a submarine for the prediction of hydrodynamic performance", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Verkuyl, J-B. and Raven, H.C., 2003, "Joint EFFORT for Validation of Full-Scale Viscous-Flow Predictions", The Naval Architect, Jan., pp. 41-43.

Wang, Z.J. and Chen, R.F., 2002, "Anisotropic Solution-Adaptive Viscous Cartesian Grid Method for Turbulent Flow Simulation", AIAA J., Vol. 40, Oct., pp. 1969-1978.

Weymouth, G.D., Wilson, R.V. and Stern, F., 2003, "RANS CFD predictions of pitch and heave ship motions in head seas", Proc. 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, Korea.

Whittaker, T.J.T., Bell, A., Shaw, M., and Patterson, K., 1999, "An Investigation of fast ferry wash in confined waters", RINA Intl. Conference on Hydrodynamics of High Speed Craft, London.

Wilcox, D., 1993, "Turbulence Modeling for CFD", La Cañada, California: DCW Industries Inc., 460 p.

Wilson, R., Carrica, P., Hyman, M. and Stern, F., 2004, "A steady and unsteady single-phase level set method for large amplitude ship motions and maneuvering", Proc. 25th Symp. Naval Hydrodynamics, St. John's, NL, Canada.

Wilson, R.V., Stern, F., Coleman, H.W., and Paterson, E.G., 2001, "Comprehensive approach to verification and validation of CFD simulations – Part II: application for RANS simulations of a cargo/container ship", J. Fluids Engineering, Vol. 123, pp. 803-810.

Wilson, R. and Stern, F., 2002, "Verification and validation for RANS simulations of a naval surface combatant", AIAA Paper No. 2002-0904.

Wilson, R., Shao, J, and Stern, F., 2004, "Discussion: criticisms of the "correction factor" verification method", J. of Fluids Engineering, Vol. 126, pp. 704-706.

Yabe, T., Xiao, F. and Utsumi, T., 2001, "The constrained interpolation profile method for multiphase analysis", J. Comput. Physics., Vol. 169, pp. 556-593.

Yamano, T., Ikebuchi, T. and Funeno, I., 2001, "Stern Waves consisting of Forward-Oriented Breaking Waves and the Remaining Following Waves", J. of Marine Science and Technology, No. 6, pp. 13-22.

Yang, Q., 2002, "Wash and Wave Resistance of Ships in Finite Water Depth", Ph.D. Thesis 2002:153, Department of Marine Hydrodynamics, NTNU Trondheim, Norway.

Yang, C. and Löhner, R., 2004, "H2O: Hierarchical Hydrodynamic Optimization", 25th Symp. on Naval Hydro., St. John's, Canada.

Zang, T.A. and Green L.L., 1999, "Multidisciplinary Design Optimization Techniques: Implications and Opportunities for Fluid Dynamics Research", AIAA 1999-3798.

Zhang, X.D., Pelletier, D., Trepanier, J.-Y. and Camarero, R., 2001, "Numerical assessment of error estimators for Euler equations", AIAA J., Vol. 39, No. 9, pp. 1706-1715.

Zhang, M., Shen, G. and Wei, R., 2002, "Error analysis of stereoscopic particle image velocimetry", Optical Tech. and Image Processing for Fluids and Solids Diagnostics SPIE-Beijing.

Zilman, G. and Miloh, T., 2001, "Kelvin & V-like Wakes affected by Surfactants", J. Ship Research Vol. 45, No 2. pp. 150-163.

12.2 Nomenclature

COMPIT	Computer and IT Applications in the Maritime Industries
FAST	International Conference of Fast Sea Transportation
HIPER	International Euro Conference on High-Performance Marine Vehicles
IWWWFB	International Workshop on Water Waves and Floating Bodies
PRADS	International Symposium on Practical Design on Ships and Other Floating Structures