# The Specialist Committee on Wake Fields

## Final Reports and Recommendations to the 25th ITTC

## 1. GENERAL

### 1.1 Membership and Meetings

The members of the Specialist Committee on Wake-Fields of the 25th ITTC are:

- Dr. Jin Kim (Chairman)
  Maritime & Ocean Engineering Research Institute (MOERI), KORDI
  Daejeon, Korea
- Dr. Thomas C. Fu (Secretary)
  Naval Surface Warfare Center Carderock Division (NSWC-CD)
  W. Bethesda, Maryland, U.S.A.
- Dr. Tomasz Bugalski
  Ship Design and Research Center (CTO-SA), Gdansk, Poland
- Dr. Munehiko Hinatsu
  National Maritime Research Institute (NMRI)
  Tokyo, Japan
- Dr. Fabio Di Felice
  INSEAN Italian Ship Model Basin
  Rome, Italy

Five committee meetings have been held during the work period:

- Daejeon, Korea, January 2006 at the Maritime & Ocean Engineering Research Institute (MOERI)/KORDI.
- Rome, Italy, September 2006 at INSEAN before the ONR Symposium.
- Tokyo, Japan, April 2007 at the National Maritime Research Institute (NMRI).
- Bethesda, Maryland, U.S.A., August 2007 at the Naval Surface Warfare Center Carderock Division (NSWC-CD).
- Gdansk, Poland, January 2008 at the Ship Design and Research Center (CTO-SA).

### 1.2 Tasks

The recommendations for the work of the Specialist Committee on Wake-Fields as given by the 24th ITTC were as follows:

- Conduct a survey of numerical methods for prediction of wake-fields at model and full scale.
- Review the experimental methods of determining the velocity distribution in the wake.
- Develop procedures for measuring the velocity distribution in the wake at model and full scale.
- Review and update the existing guidelines for the simulation of the wake-field for cavitation testing.

## 2. INTRODUCTION

The recommended actions of 25th ITTC Specialist Committee on Wake-Fields, as stated above are focused on two main areas, the review of the numerical prediction and experimental measurement (methods) of wake-fields and the review and development of ITTC procedures. Since the Committee on Wake-Fields is a Specialists Committee the literature survey of numerical and experimental methods was not constrained to the period since the 24th ITTC Meeting, and the emphasis of these tasks was on completeness. Much attention was paid to the numerical prediction of wakes at full-scale, as this has been a major research topic in recent years. The review of experimental methods includes efforts at both model and full scales and also focused not only on measurements for simulated wake verification but also for Computational Fluid Dynamics (CFD) validation and basic research. This committee's tasks related to the review of existing ITTC procedures and the development of procedures, and was, as expected, more focused, concentrating on the generation, use, and verification (measurement) of simulated wakes.

By reviewing all of the ITTC Procedures for references to wake-fields, it was found that only four existing procedures refer to wake-fields. These procedures are:

- Procedure 7.5-02-03-03.1: Model-Scale Cavitation Test Cavitation Induced Pressure
- Procedure 7.5-02-03-03.3: Fluctuations Model Scale Experiments
- Procedure 7.5-02-03-03.5: Cavitation Induced Erosion on Propellers, Rudders and Appendages Model Scale Experiments
- Procedure 7.5-02-03-03.6: Podded Propulsor Model-Scale Cavitation Test

Review of these procedures was made to standardize the procedures for generating and assessing simulated wakes and these modifications have been submitted.

Development of wake-field measurement procedures utilizing 5-Hole Pitot Tubes, Laser Doppler Velocimetry (LDV), and Particle Image Velocimetry (PIV) for both model and full scale were all considered by the committee. The committee assessed the current status of full-scale wake-field measurements and recommended to the Advisory Committee that procedures for full-scale measurements not be pursued, because these types of measurements are far from routine. Similarly, PIV, while widely utilized to measure velocity fields, is not commonly utilized to measure nominal wake fields at the propeller plane, because of issues related to optical access (obstruction). Additionally, PIV is typically utilized as a research tool and the committee felt it was premature to establish procedures for its use. For these two reasons, the committee recommended that PIV procedures not be developed, though guidelines for the use of PIV are provided in Section 7.0 of this report.

Procedures for nominal wake field measurements utilizing 5-Hole Pitot tubes and LDV were developed and submitted to the Advisory Committee. Additionally, guidance for generating simulated wake-fields for model scale testing is given in Section 6.0.

What follows are sections on the review of the numerical predictions (Section 3), of experimental methods of wake-field measurements (Section 4), followed by section 5 on the developed procedures on the use of 5-hole Pitot Tubes and LDV to measure nominal wakes, and finally sections on the reviewed procedures (Section 6), guidance on the use of PIV (Section 7), and the committee's recommendations (Section 8).

# 3. SURVEY OF NUMERICAL METHODS FOR PREDICTION OF WAKE FIELDS AT MODEL AND FULL SCALE

## 3.1 Introduction

Since the 24th ITTC, a number of advances in numerical methods have occurred leading to improved predictions of the wake-field. This report includes a survey of the papers published on both global and regional conferences, journals, benchmarkings and other types of scientific activities. Further open results of EFFORT and LEADING EDGE European Union joint research projects were taken into account.

December 2005 saw the successful conclusion of the three-year European Full-scale FlOw Research and Technology (EFFORT) project, a 5th Framework EU project. Principal objective was to validate CFD predictions of viscous flow and wake field of ships by comparison with experimental data for full-scale. Such data being scarce, new measurements of wake-fields of two ships at full scale have been done. Full-scale flow measurements were conducted by MARIN, for 'Nawigator XXI', a single screw research vessel of the Maritime University of Szczecin, and 'Uilenspiegel', a hopper dredger of Dredging International; a twin-screw vessel with bossings, exposed shaft hoses, shaft support struts and ducted propellers. A dedicated LDV system was used, operating through windows flush-mounted in the hull. As usual, these measurements were not without problems, due to environmental conditions and other complications; but the data obtained form a valuable addition to existing material. Parallel model experiments were carried out, including Pitot measurements by HUT, PIV measurements by CTO and wave pattern measurements using a laser sheet technique by NTUA. On the CFD front, RANS computations were requested for full-scale Reynolds numbers. For some methods this was a challenge in itself, the extreme grid density at the hull surface causing problems with convergence or stability. However, most codes passed the test and could be further extended and applied. Developments were done on modeling the propeller action, comparing turbulence models and including the wave pattern. An internal EFFORT workshop compared results and stimulated improvements of the accuracy and applicability of several codes. All model and full-scale experimental data, for the two new ships and five existing ones, were then used to study the achievable level of accuracy of the predicted wake fields at full-scale, to decide on the best turbulence models and to find out desired extensions of the methods. Finally, the codes were used in case studies for practical applications, in cooperation with the industrial participants.

Several of the CFD codes were found to predict the full-scale flow relatively accurately, provided fine enough grids were used. Already some further improvement has been achieved by a changing turbulence models; while for a case like the dredger, incorporation of propeller ducts is a desired step. The full-scale predictions for practical projects, already frequently made, thus may play an even larger role soon. EFFORT certainly lived up to its name; ambitious goals were set at the start and not all could be achieved. But definitely an important contribution has been made to the practical use of CFD for prediction of the full-scale viscous flow and propeller action.

The demand of ship owners for efficient and quiet operation of ships at relatively high speed or at high power is a challenge for the shipbuilding industry. The propeller plays herein a significant role and is, for instance, a critical element in the control of vibrations and noise on board. There is, however, a notable lack of tools in predicting and scaling the development and evolution of the vortex at the tip of the propeller blades. The LEADING EDGE project aimed at providing such tools to industry (shipyards, model basins, propeller designers and manufacturers).

To that end, the knowledge of and experience with computational codes and techniques (RANS codes) at several universities and institutes in Europe has been brought together and applied to ship propellers. The results obtained give information about the origin, the strength and the structure of the non-cavitating tip vortex at model and at full scale. Complementary experiments have been carried out, among others with the latest PIV techniques. Transfer of the tool to industry has been given support via the execution of an optimization process, led by industry, in which the results of the new tool were related with existing experience in industry.

Three representative propellers were chosen for this project, viz. a conventional propeller (used as a reference propeller), a modern high-skew propeller and a tip-plated propeller. The propellers have been selected to represent the typical features of leading edge and tip vortex cavitation. The highly skewed propeller (comparable to a swept aircraft wing) characteristically shows the development of a vortex along a significant part of the leading edge, before changing into a tip vortex. By fitting a tip plate (comparable to wing tips on airplanes) the vorticity accumulation in the tip vortex may be beneficially influenced.

The first step in the project was the development of an exchangeable geometry, to ensure that all partners do their calculations for exactly the same geometry. The resulting geometries are available in IGES neutral format for information exchange between partners and at the same time as benchmark data for later use and as examples of the generation of the propeller geometry from sectional data.

Eight participants have calculated the flow around one or more of the propellers, at model scale as well as at full scale. A detailed inter-comparison study has been carried out. Results of validation measurements at model scale have been obtained. These measurements include traditional propulsion, open-water and cavitation inception tests, but also flow field measurements with 3D Particle Image Velocimetry (PIV), a technique which has been applied in ship model basins in the past only incidentally, but recently is increasing in use. Based on the results of measurements and computations an attempt at blade shape optimization has been made for all three propellers. The new designs have subsequently been investigated numerically.

The LEADING EDGE project has shown that viscous flow analysis for ship propellers is feasible and useful. A better understanding of the tip vortex development and the scale effects thereon has been obtained.

All “wake-field” papers found during survey are listed in Literature. Some of most significant of these papers are discussed in this report.

### 3.2 Viscous Flow Computations at Model Scale

The 2005 CFD Workshop Tokyo [1] included wake computations for KRISO container ship (KCS), KRISO tanker ship (KVLCC2M) and DTMB 5415 combatant ship. Theses efforts provide examples of the current CFD capability to simulate the viscous wake-field around a ship. For viscous flow computations, almost all contributors solved the RANS equations. Turbulence models were either two-equation models such as k-$\varepsilon$, k-$\omega$, or combined models or 1-equation model such as Spalart-Allmalas type, algebraic turbulence models and Reynolds stress models.

Rhee & Stern (2001) [2] reported results of an unsteady Reynolds-averaged Navier–Stokes (RANS) method for simulation of the boundary layer and wake and wave field for a surface ship advancing in regular head waves, but restrained from body motions. Second-order finite differences were used for both spatial and temporal discretization and a Poisson equation projection method is used for velocity–pressure coupling. The steady flow results are comparable to other steady RANS methods in predicting resistance, boundary layer and wake, and free-surface effects. The physics of the unsteady boundary layer and wake and wave field response are explained with regard to frequency of encounter and seakeeping theory.

Alin et al. (2003) [3] presents results from a computational study using Large Eddy Simulation (LES) of the flow past a fully appended submarine hull in model scale (Re=1.2·107). The aims are to evaluate LES for high-Re hydrodynamic flows, to investigate the influence of subgrid modeling and spatial grid resolution, and to discuss some generic features of submarine hydrodynamics. Results are presented for the towed baseline case, for which experimental data are available for comparison, and for a corresponding self-propelled case. Good agreement is found between the experimental and the LES results.

Bensow et al. (2006) [4] investigated a comparative study of RANS, DES and LES. High Reynolds number wall bounded flow was calculated using Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and Reynolds Averaged Navier Stokes (RANS). The one of case is a flow past an axisymmetric submarine hull with an elliptic forebody and smoothly tapered stern. For the axisymmetric hull RANS and DES show reasonable results but somewhat inferior when compared to the LES.

Bugalski & Kraskowski (2006) [5] presents validation of the RANSE wake computations for the training vessel Nawigator XXI and the dredger Uilenspiegel – result of work for EFFORT project.

### 3.3 Nominal Wake

In the CFD workshop Tokyo 2005 [1] many cases related to the computation of wakes were presented. These were:

1) Wake simulation of KCS with free surface effect, under fixed trim, and sinkage condition, referred to as test case 1.1.

2) Wake simulation of DTMB model 5415 with free surface effect, under fixed trim and sinkage condition, referred to as test case 1.2.

3) Wake simulation of KVLCC2M model without free surface effect, under fixed trim and sinkage condition, referred to as test case 1.4.

4) Wake simulation of KVLCC2M under obliquely towed conditions, without free surface effect, under fixed trim and sinkage condition, referred as test case 3.

5) Wake simulation of KVLCC2M of test case 1.4 with 5 different grid mesh fineness, referred to as test case 5.

On the wake simulation for KCS model, test case 1.1, a total of nine organizations made predictions and eleven numerical results were presented. As summarized in the workshop proceedings, most of the estimated axial wake patterns, as well as the patterns of computed cross flow vectors, agree well with the experiment. However, for the velocity distributions on a horizontal line, large discrepancies were found in axial velocity profiles between the experiment and the computations, particularly near the centerplane.

In the wake simulation of DTMB 5415, nine organizations presented eleven numerical results. As similar to the case of KCS model, numerical estimation of the wake pattern was in good accordance with the experiment, because DTMB 5415 hull is also a fine ship like KCS and no strong longitudinal vortices exist around the stern of the ship.

For the wake simulation of KVLCC2M forward speed condition, nine organizations showed a total of thirteen computations. (Test case 1.4) For the wake simulation around the full ship hull, reproduction of the "hook" shape in the axial wake pattern was a focus. Some results were able to resolve the hook shape patterns while some could not. Although the same turbulence model was used, reproduction of the hook shape depended on the CFD code. Results were also introduced that included a modification of the Spalart-Allmaras turbulence model to the Modified Spalart-Allmaras model, which did give the "hook" shape wake pattern. It seems to still be a difficult problem to simulate the hook shape wake for a full-hull ship. On the velocity profile along a horizontal line, large discrepancy was still found near the centerplane of ship. In the workshop, grid dependency was also investigated. For the CFD codes that could reproduce the hook shape, the finer the grid is the slightly sharper the hook pattern became. This grid dependency test should continue to be used to develop and revise CFD codes.

For the obliquely towed conditions, nine organizations took part in the simulation of flows and a total of 16 and 15 results were introduced for oblique towing angles of 6 and 12 degrees, respectively. Each of the simulated wake patterns is very similar and agrees well with the experiment. However the wake domain became wider compared to the experiment, due to the coarseness of grid.

### 3.4 Total or Effective Wake

At the CFD Workshop Tokyo 2005 [1], four institutions contributed to the topic of total or effective wake. They showed the comparison of the computed wake distribution behind an acting propeller for KCS model with free surface effect, under fixed trim and sinkage condition, referred as test case 2 in the workshop. As mentioned in the summary in the workshop proceedings, Simulated cross flow velocity distributions for each computation agreed reasonably well with the experimental data. The asymmetry in the axial wake pattern was also found in some computational results. This is a typical feature of the wake pattern behind an acting propeller. The velocity distributions along a horizontal line showed some discrepancies between the computations and experiments. As one of the reasons for such discrepancies, the Workshop pointed out that propeller models do not take into account effects of the propeller hub.

The European-Union EFFORT (European Full-Scale Flow Research and Technology) project focused on validating and introducing innovative Computational Fluid Dynamics (CFD) prediction methods for the performance of the ship/propeller combination at the full scale, instead of the usual model scale. Within the project, existing CFD codes have been extended, extensive full scale and model-scale (PIV) flow measurements have been conducted, and validation of the CFD predictions have been carried out. Several papers (Visonneau, Starke, Verkuyl, Bugalski…) provide an overview of the EFFORT project results and the innovations that have been implemented by the EFFORT consortium group. The model scale PIV experiments that have been carried out on two vessels are also discussed. In the papers there are shown some of the validation results based on these experiments. Finally there are selected examples of the calculations that have been carried out by EFFORT partners, in which the validated CFD computations have been used as a design tool replacing the model tests which are normally carried out.

Hino (2006) [6] describes a performance estimation system based on CFD methods. For efficient use of CFD, various systems components are introduced. Through an application example, the usability of the approach is demonstrated and evaluated by comparison with the measured data.

Kim et al. (2006) [7] presents hybrid RANSE and potential based numerical simulation of a self- propulsion test for a practical ship. The paper shows that hybrid RANS and potential flow based numerical method is promising to predict the self-propulsion parameters of practical ships as a useful tool for the hull form and propeller design.

Kimura et al. (2007) [8] shows a study on interaction between a ship hull and propeller. The RANS solver is coupled with lifting surface Vortex Lattice method for the highly accurate computation of propeller flows. The propeller effects are integrated in RANS equations by the body forces equivalent to mean forces acting on propeller blades. The authors discuss the hull-propeller interaction, regarding the self-propulsion factors and the influence of wake distribution without/with propeller.

### 3.5 Rudder/Appendages

Modern ship hulls have a complicated shape and their geometry becomes more complex by the presence of propulsors, rudders and various appendages. Therefore, ship hull design tools should be able to cope with bodies of complex geometry. Due to its flexibility of handling such complex geometry, unstructured grid methods are expected to be one of the most powerful CFD approaches for use of practical design applications. It is also important to estimate not only the resistance of a given ship hull but also its propulsive performance. In order to do this, propulsor effects should be taken into account in the numerical simulation procedure.

Hino et al. (2006) [9] presents an unstructured grid based Navier-Stokes solver with a body force model for propeller effects and applied it to the flows around unpropelled and self-propelled ships with appendages. Numerical results are compared with the experimental data.

Laurens and Cordier (2003) [10] presents numerical simulation of the propeller-rudder interface. The obtained results are in good agreement with available experimental results. Moreover, the computational speed allows a parametric analysis of the interaction (rudder position, propeller load, etc.) to be carried out.

In order to determine the dimensions of the rudder, a reasonably accurate prediction of the hydrodynamic forces generated by this configuration should be made, otherwise the traditional approach which consists of replacing the operation of the propeller by an actuator disk only provides a reasonably accurate estimate when the propeller load is relatively low. On the other hand, a direct simulation of the interaction by using a Reynolds-averaged Navier-Stokes (RANS) calculation demands, in addition to a large number of computer tools, a major effort in preparing the grid which cannot be used for many practical applications or parametric studies unless sufficient time and appropriate computer tools are available. Several methods were developed to simulate this interaction by coupling the potential code and the RANS code. The best results are obtained by using the velocity field calculated in the near-field downstream flow from the propeller simulated by the resolution of the potential problem as inlet boundary conditions within the RANS calculation. After the RANS simulation has been carried out, the calculated velocities in the RANS domain are interpolated to the control points of the rudder. A simulation of the rudder is then carried out by resolving the potential model while taking the distortion of the flow due to the presence of the propeller into account.

Kim et al (2006) [7] presents numerical computation of stern flows around various twin skegs geometry of ship hull. Numerical analysis was conducted according to the variations of distance between skegs and vertical skeg inclinations by using a WAVIS hydrodynamic analysis system. It shows that numerical analysis can be useful for the complex stern hull forms with twin skegs.

Regnstrom & Bathfield (2006) [11] presents drag and wake prediction for ships with appendages using an overlapping grid method. A flexible and robust overlapping structured grid method is introduced and applied to the computation of flow around two ship hulls with appendages. The ships are a frigate with sonar dome, bilge keels, propeller shafts, brackets, nozzles and rudder and a hopper–dredger with head–box, shafts, brackets and nozzles. Results are presented both for model and full scale and compared to experimental data. The work was done for the EFFORT project.

Bensow et al. (2006) [12] shows propeller near wake analysis using LES. LES on a rotating mesh has been applied for the simulation of the flow around a propeller, focusing on an investigation of the velocity field in the near wake. The objective is the validation of LES for the purpose of hydrodynamic and geometrical characterization of the wake flow field and its downstream evolution.

### 3.6 Viscous Flow Computations at Full Scale

Successful CFD prediction of the full-scale viscous flow and propeller inflow are a large step forward compared to the current practice of model-scale testing or computations, followed by empirical scaling methods. Reduced resistance, improved propeller inflow and propulsive efficiency, and reduced cavitation and vibration can be achieved. With the use of CFD, hull form optimization can be done in approximately 3 months, while approximately 6 months would be needed for model tests. Moreover, CFD tends to give better clues to how and where improvements can be obtained. Accurate full-scale CFD predictions will contribute to early decision making in the design process, which can have a strong impact on the costs later. Short delivery times can be met, which helps in bidding processes.

Tahara et al. (2003) [13] performed numerical simulations of a full-scale ship flow utilizing RANS equations for the governing equations. Additionally, roughness effects were included in the flow model. Computed results were also compared with Tanaka’s correction method.

The European-Union project EFFORT (European Full-Scale Flow Research and Technology) focuses on the applicability of different RANS methods to full-scale viscous flow computations.

Schweighofer (2004) [14] investigates the turbulent free-surface flow around the Series 60 ship at model- and full-scale ship Reynolds numbers. The steady RANS equations are solved using the RANS solver FINFLO SHIP developed at the Ship Laboratory of Helsinki University of Technology. Turbulence is treated with Chien’ s low-Reynolds number k-ϙ turbulence model. The significance of computations and validation at full scale is discussed. The results shown demonstrate that free-surface computations of turbulent ship flows around simple geometries are possible at full-scale ship Reynolds numbers using the moving-grid technique and no wall functions.

Visonneau (2005) [15] presents studies carried out in one of the major EFFORT project work-packages which had three main objectives: (i) to develop and implement the appropriate physical modelling for full scale flows,(ii) to perform numerical studies of full scale flows around a real ship to check the robustness and the accuracy of the simulation tools in full scale flow conditions, (iii) to issue recommendations to prepare the simulation tools to be used for CFD validation at ship and model scale, to develop and implement the appropriate physical modelling for full scale flows. Visonneau et al. (2007) [16] shows application of the developed method to simulation of the full scale flow on two appended hull configurations.

In the paper of Schweighofer et al. (2005) [17], first results are presented with respect to the validation of the different approaches used. The viscous flow around two existing vessels was computed at model-and full-scale Reynolds numbers with and without acting propeller. The free surface was taken into account in both cases. Depending on the approach, the boundary layer was either resolved till the wall or wall functions were used. The used approaches and computations are briefly described. An evaluation of the grid influence on the results is given. The convergence criterions used are given.

Results, mainly related to the wake, are presented and compared with measurements as far as permitted. The application of numerical methods to viscous ship-flows including the propeller action is discussed. The studies have shown that at full scale, there is a significant effect of the turbulence modelling on the predicted wake field. In some cases, the results with respect to the evaluation of the wake are in very good agreement with the experimental ones indicating that the respective numerical methods may be used for the accurate prediction of the model- and full-scale wake fields of similar ships.

Starke et al. (2006) [18] presents and discusses full-scale wake field validations for a number of ships. It is shown that accurate viscous-flow computations are possible at full scale, which shows good correlation with experimental data measured during sea trials with ship mounted LDV systems. It is shown that the effect of turbulence closures on the longitudinal vorticity, well-known at model scale, occurs at full scale as well. Of the turbulence closures tested, those that perform well at model scale are also found to perform well at full scale.

Verkuyl et al. (2006) [19] presents an overview of the EFFORT project results and shows the numerical developments that have been implemented by six CFD European groups. It is also discussed the full scale LDV measurements and the model scale experiments that are carried out on two vessels. The paper shows some of the validation results based on these experiments. Finally the paper shows selected examples of the application and demonstration cases that were carried out with the industrial partners in which the validated CFD computations have been used as a design tool replacing the model tests which are normally carried out.

Han et al. (2007) [20] presents numerical optimization of the propeller behind a ship hull at full scale. A propeller is optimized to minimize the delivered power at a given ship speed by adjusting the geometry and the revolution speed. An uncertainty analysis and a wake validation are made for the SHIPFLOW code.

### 3.7 Conclusions

The survey of numerical methods for wake field predictions presents the rapid development of these tools. It is shown that numerical methods for prediction of wake fields can be useful for the complex stern hull forms at the model and full scale and a variety of rudder/appendages configurations, etc.

For wake field computations almost all scientists solve RANS equations. Reynolds Stress Model (RSM) and 2-equation model as k-$\varepsilon$, k-$\omega$, or their combined models are mainly used as a turbulence model.

The EFFORT project shows that RSM turbulence model is the most reliable for wake field calculations.

## 4. REVIEW OF EXPERIMENTAL METHODS OF DETERMINING THE VELOCITY DISTRIBUTION IN THE WAKE

### 4.1 Introduction

In the following section, a review of the experimental techniques used in towing tanks for wake flow analysis is reported. The goal is not to introduce detailed information on the principle of the measurement techniques, but instead, to provide a picture of the possible applications and capabilities in towing tank testing. Furthermore, highlights on the future development on velocimetry techniques will also be discussed.

## 4.2 Measurement Techniques

There are several different methods which can be used to measure the ship wake. A general classification of these methods is based on the capability of the measurement techniques to measure at a point, in a plane, or in a volume. Table 1 is a comparison of these velocimetry techniques

### 4.2.1 Single Point Techniques (Pitot Tube, LDV)

Pitot Tube, Laser Doppler Velocimetry (LDV), and Hot Wire Anemometry (HWA) are typically single point measurement techniques that have the ability to measure up to three velocity components. With the exception of HWA, which is rarely used and is mostly suited for investigating turbulent flows, single point techniques are the most common method in tow tank testing for nominal wake measurements.

#### 4.2.1.1 Pitot Tube

One of benefits of using a 5-hole Pitot Tube is the ratio between cost and performance to measure the three components of velocity simultaneously. Both pressure transducers and a traverse system are common tools for tank experiments. Since the principle of analysis is based on potential flow theory, it is a clear choice to use this method for flow measurements.

There are however, some disadvantages to this method. The Pitot method gives velocity indirectly through the measurement of pressure and from the principle of measurement, the technique is not linear. Additionally, it is difficult to extend to the unsteady flow field, and it's likely that the flow field will be disturbed by the measurement apparatus. However, in spite of these issues, the Pitot method is commonly used in towing tank and wind tunnel experiments and remains an adequate technique to measure the steady flow field.

To date, many of the issues related to 5-hole Pitot have been addressed; examples of these works are introduced here. Pien (1958) [21] introduced an analysis method of flow measurement through the use of a 5-hole Pitot tube. This method is based on the potential flow around a sphere and where horizontal and vertical velocity components are determined independently. This is done through pure horizontal and vertical calibration curves, an analysis method known as the one-dimensional calibration method. Wright (1970) [22] proposed another analysis method within the same category.

A one-dimensional method may however lead to a large measurement error when the vertical and horizontal inflow angles have simultaneous large values. To avoid this deficit, Fujita (1979) [23] and Olivieri (2003) [24] presented a new calibration method in which the Pitot calibration is carried out in two-dimensional horizontal and vertical angles, a method which is known as the two-dimensional calibration method. Although this two-dimensional method requires more time to create a calibration chart compared to the one-dimensional method, the new method is expected to create more accurate results in comparison. For this reason, the two-dimensional method would be the preferred method for the analysis of Pitot measurements.

#### 4.2.1.2 Laser Doppler Velocimetry (LDV)

Of the various techniques based on light scattering, Laser Doppler Velocimetry (LDV) is the most common and well-established method. LDV has the ability to measure three components of the velocity vector at a single point as a function of time. This measurement point can be moved in the flow field using a traversing mechanism, to obtain complete velocity fields. Among the single point measurement techniques, it is unique in that it

| | measured components | measurement location | frequency resolution | data amount per s (Mb) | processing time (s) | note |
|---|---|---|---|---|---|---|
| **5 holes Pitot tube** | 3 | single point | up to 10-100 Hz | < 0.01 | real time | Requires priori knowledge of the flow direction. Not suitable in detached and reverse flow |
| **LDV** | 3 | single point | up to 5-10 KHz | 0.02 | real time | Able to resolve flow direction |
| **PIV** | 2 | plane | up to 30 Hz | 64 | 240 | Time for processing evaluated on a Xeon processor 2.7 GHz, with a 2 step multi-grid algorithm |
| **SPIV** | 3 | plane | up to 30 Hz | 128 | 520 | Time for processing evaluated on a Xeon processor 2.7 GHz, with a 2 step multi-grid algorithm. Stereo reconstruction based on Solo method |
| **Time Resolved PIV** | 2 | plane | up to 5-10 KHz | 4000 | 7500÷30000 | Minimum time for processing based on direct analysis of a image pairs. Maximum processing time based on a multi-frame analysis of 4 images |
| **Time Resolved SPIV** | 3 | plane | | 8000 | 16500÷66000 s | |
| **DDPIV** | 3 | volume | up to 30 Hz | 192 | 600 s | |
| **Time Resolved DDPIV** | 3 | volume | up to 5-10 KHz | 12000 | 20000-80000 s | |

Table 1: Comparison of different velocimetry techniques

is able to resolve the direction of the flow and has been shown to be suitable for complex detached and reversed flow analysis.

LDV is now routinely used by many research organizations throughout the world, because it provides valuable information on the mean and fluctuating velocity field in complex situations. Measurements by LDV have proven to be a turning point in technology because of the ability to analyze the three-dimensional complex flow in ship wake field around rotors and propellers, which provides very valuable flow data for CFD validation.

The critical concern in using LDV focuses on the single point nature of the technique. More specifically, drawbacks involving both technical and logistical aspects are listed hereinafter:

- It is difficult for LDV to provide the spatial characteristics of the large scale eddies that are encountered in complex and separated flows.
- The Eulerian and time averaging nature of the technique causes significant errors on the intensity estimation of unsteady vortical structures.
- Long period facility operations are required to achieve a whole velocity field with adequate spatial resolution. This causes an increase in testing costs.

In regard to the increased testing costs, the benefit of using LDV, (i.e. the ratio between the time to measure one plane and the acquisition-time-per-point), primarily depends on the type of facility. Other variables do contribute to these costs, such as the traversing speed and the point-to-point upload time of the LDV processor.

The efficiency of LDV decreases for facilities in which data acquisition cannot be performed continuously. For example, this can occur in towing tanks in which the acquisition time is limited by the duration of the towing carriage run; the pauses between consecutive runs have to be planned to assure the complete damping of the waves induced by ships/models. A useful recommendation to remedy this situation focuses on setting the grid scanning order properly in order to minimize the point-to-point displacement of time and thereby maximizing the number of points measured during one run of the towing carriage. In addition, an approach in which grids are refined iteratively by adding points where the smaller structures are resolved by previous steps is suggested for applications in which the location of the flow structures are unable to be predicted with sufficient accuracy.

In recent years, LDV literature focuses mainly on the propeller/waterjet wake survey and the propeller-hull interaction. Moreover, LDV measurements with phase sampling techniques are reported in the most recent literature from David Taylor Model Basin and INSEAN.

The propeller wake survey in behind condition was executed by Felli and Di Felice (2005) [25] in the large cavitation tunnel of INSEAN using LDV and phase sampling. Both the inlet to the propeller and the downstream wake were measured using a 2C-LDV probe. In the former, the lack of optical access, due to the propeller shaft, required two optical configurations with one laser beam coming from below and another from the side. This provided for the ability to obtain the axial component of the velocity along the upstream propeller plane and the 3D wake only in the regions reached by both configurations. The downstream wake was acquired in two steps by measuring the cross flow and the velocity components in the vertical plane.

LDV phase sampling techniques were also used by Felli and Di Felice (2002) [26] to measure the three-dimensional velocity field in transversal sections of a twin screw ship model, located just downstream of the sonar dome, of the bilge keels and of the fin stabilizers in towing tank experiments.

Felli and Di Felice (2004) [27] completed the study measuring the propeller-hull interaction just upstream of the propeller and behind the rudder. In both experiments, measurements were obtained in the towing tank of INSEAN using a two component LDV system and unevenly spaced Cartesian grids of about 400 points.

The propeller-rudder interaction was also studied by Felli et al. (2006) [28] in the cavitation tunnel of the Italian Navy, using a 2C probe and phase sampling. In this case, the LDV probe was moved along two transversal-Cartesian-grids of about 600 points, just in front and just behind the rudder.

LDV measurements were made in four sections of a waterjet pump by Micheal and Chesnakas (2004) [29] in the water tunnel of NSWCCD. The following approaches were dependent on the local features of the flow and the nature of the window access in the facility: 1) radial grid and phase sampling in the flow regions influenced by the rotor perturbation (i.e. rotor inter-blade section and section in between the rotor and the stator); 2) radial grid and standard LDV techniques at the rotor inlet where the flow was assumed axisymmetric, 2D Cartesian grid and 3) standard LDV techniques at the nozzle exit, where the flow was primarily influenced by the stator and remained non-axisymmetric.

Jessup et al. (2004) [30] studied the propeller wake features in extreme off-design conditions in the water tunnel of NSWCCD. The three directions of movement in the flow field were provided through axial and radial translations of the LDV probes and phase sampling.

Felli et al. (2004) [31] investigated the effect of the ship appendages in roll damping by testing the bare hull and fully appended configuration of the DTMB 5415 in the large cavitation tunnel of INSEAN. Tests were performed in a forced roll condition along eight cross sections of the ship model and, as a result, roughly 6000 points were acquired. The measurement phase was performed locked to the roll angle and a particular procedure was devised to achieve the medium between the randomly acquired LDV samples and the roll angle of the ship model. This was obtained by feeding the frequency modulated signal of the roll angle to a third LDV processor. This was done in order to acquire the ship position, the velocity signal time histories, and to reconstruct the phase in post-processing.

## 4.2 Planar (PIV, SPIV, Underwater PIV probes)

### 4.2.1 Particle Image Velocimetry (PIV) and Stereo Particle Image Velocimetry (SPIV)

The strong development of lasers, fast digital cameras and computers in the last ten years has allowed decisive progress in velocity field methods derived from visualization techniques; PIV is an example. The flow seeded by micron particles is illuminated by a sheet of laser light and two images of each particle in the sheet are recorded at a short time interval on digital high resolution cameras. The measurement of the distance separating the two images of each particle provides a local velocity vector. More details about the techniques can be found in Raffel et al. (2007) [32].

In contrast to LDV, PIV has many features that make it popular. First, PIV makes it possible to study the spatial structure of the flow because it is a whole-field measurement technique that maps full planar domains. The recording time is also very short and depends mainly on the technological characteristics of the acquisition and processing components, for which the analysis can be done a posteriori. The most definitive advantage is when high-cost facilities are being used; PIV operation is less demanding in terms of technical expertise.

Stereoscopic PIV (SPIV), the latest evolution of planar PIV, allows three-component velocity measurements in a plane through the use of two cameras that image the flow field from two different directions. Each camera measures an apparent displacement perpendicular to its optical axis. By combining the information from both views, it is possible to reconstruct the actual three-dimensional displacement vector in the plane. In naval hydrodynamics, PIV has been widely expanded in recent years since its first application in towing tank experiments by Dong and Katz (1997) [33]. These authors used PIV on underwater vehicle flows, ship wake, ship roll motion, propeller flow, wave breaking, freak waves, free surface flows, bubbly flows, boundary layer characterization for drag reduction by microbubbles, sloshing flows, and waterjets. Due to its many applications, and testing facility environments, specific developed instrumentations and applications, the following will focus on propeller and ship wake applications.

PIV, due to its ability to capture the instantaneous flow field at a plane, is an important tool for the analysis of the propeller wake coherent structures like the tip vortices which dominate the slipstream. The presence of strong velocity gradients and velocity dynamics in the propeller wake drove Cotroni et al. 2000 [34] to improve PIV image analysis, introduce iterative multigrid and create window deformation algorithms. Today, these practices are widely used.

Di Felice et al. (2004) [35] and Felli et al. (2006) [36] used PIV in a cavitation tunnel to study the main characteristics of the propeller wake at different loading conditions by using phase-locked measurements. After, the longitudinal evolution of the blade viscous wake, of the trailing vortex sheets and tip vortices were analyzed. The interaction between the different coherent flow structures was examined as were the development of the slipstream instability and the breakdown of the hub and tip vortices. Paik et al. (2007) [37] performed a similar study which focused mainly on the analysis of the tip vortex trajectory up to one propeller diameter.

Stereo-PIV was first applied in the naval field by Calcagno et al. (2002) [38] in a large cavitation tunnel. Measurements were performed on a 60-series ship model with a MAU propeller installed. An important study of the propeller in crash-back condition was performed by Jessup et al. (2004) [39] and (2006) [40]. PIV and Stereo PIV have been applied to characterize the propeller flow in an effort to provide valuable data for RANSE validation.

Recently, dedicated underwater PIV probes have been developed for towing tank applications. Figure 1 depicts the first underwater PIV system for towing tank applications used by Gui et al. (2001) [41]. This system, for the limited energy of the laser and camera resolution, is able to perform measurements on the order of 100 x 100 mm$^2$.

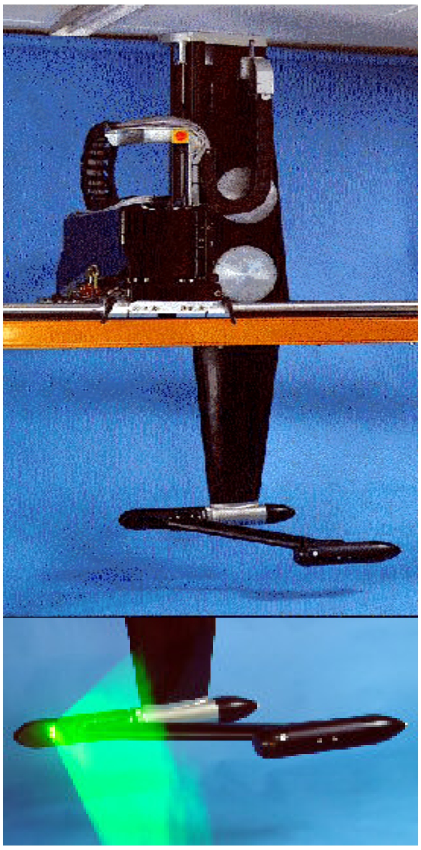

**Figure 1: First underwater PIV probe (Gui et al 2001)**

INSEAN developed the first underwater Stereo-PIV system (Figure 2) for towing tank applications. The system has a high energy laser, four megapixel cameras and is suitable for the measurement of the three velocity components. The measurement capability is up to 400 x 300 mm$^2$ and the system can be configured in several ways to accommodate testing needs. More details can be found in Felli et al. (2003) [42] and Di Felice and Pereira (2007) [43].

Underwater probes for towing tank applications are growing steadily. Within the European Hydro-Testing Alliance, a cooperative effort and joint research venture has begun for the development of standards on PIV methods in towing tanks. Due to the wide range of possible applications and difficulties related to hardware and calibration methods, these standards are still in the early development stage.

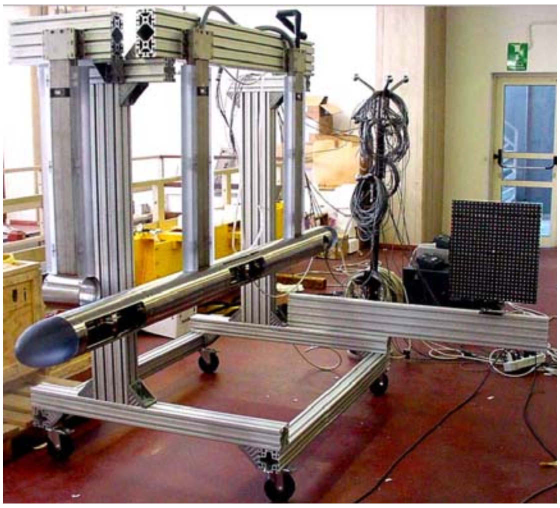

**Figure 2: First underwater StereoPIV Probe (Calcagno et al 2003)**

## 4.3 Volumetric (tomography, DDPIV)

Our review suggests that the new technological challenges, posed by applications of ever-increasing complexity, are constantly motivating further progress on both the hardware and software. This also suggests that the trend observed over the past several years is moving decisively toward the full three-dimensional flow-field characterization. This trend has motivated the development of leading-edge technologies that have yet to be validated on real cases. Despite this, flow three-dimensionality is becoming a common feature in ship and propeller wake because the three component, two-dimensional information provided by a standard stereoscopic PIV approach, can only provide a partial view of the global problem.

The current efforts in the PIV community are related to the extension of the PIV concept to the full three-dimensional space. A number of interim solutions have been developed, such as the multilayer PIV [44], scanning PIV [45] and multiplane PIV [46]. Currently, true volumetric information for general fluid dynamics research is attainable through the use of holographic PIV [47], particle tracking velocimetry [48], tomographic PIV [49] and defocusing PIV [50].

Holographic techniques such as HPIV have been praised for their high potential but have reluctantly been implemented in non-laboratory environments due to their delicate optical configuration and sensitivity to external perturbations. The optical reconstruction of the 3D field from the recorded hologram allows for post-interrogation by techniques such as microscopic photography, PIV or PTV. In bubbly flows it is possible to also obtain population and size distributions of cavitation nuclei HPIV [51] and [52]. However, the interrogation process of the holograms is a time-consuming operation and still relies heavily upon human supervision, even though automatic procedures are now available.

PIV provides an Eulerian representation of the flow’s physical descriptors. In contrast, the Particle Tracking Velocimetry (PTV) technique is a true three-dimensional approach providing a more natural description known as Lagrangian representation. PTV is derived from traditional flow visualization techniques like streak photography, and is based on the coordinate determination by triangulation and on the tracking of individual markers, such as bubbles or neutrally buoyant seeding tracers.

Particle tracking is usually applicable under two restrictive conditions. First, the concentration of scatterers must be low in order to perform a reliable identification. With large densities, ambiguities arise and tracks cannot be reconstructed correctly, unless redundant information is made available by additional recording cameras. Second, the displacements, or tracks, must be small enough when compared with the mean spacing between the particles. As an example, if PTV overcomes the incapacity of PIV to follow the motion of particles, this will be done at the price of a lower spatial resolution for the velocity field.

Tomographic PIV bases the reconstruction of the volumetric particle field upon the MART algorithm. It requires imaging from multiple observation locations, which restricts the application to small scale laboratory flow cases.

The defocusing digital particle image velocimetry (DDPIV) technique is a new approach to the three-dimensional mapping of flow fields which naturally extend the planar PIV techniques to the third spatial dimension [53]. DDPIV was designed to map relatively large volumes (in the order of one cubic-foot) and is based upon an optical concept that allows the instrumentation to be self-contained.

The interrogation domain is a volume where three-dimensional coordinates of fluid markers are determined prior to flow analysis. However, unlike PTV or stereo-based methods, DDPIV has one unique optical axis and is based on pattern matching rather than on stereoscopic matching of particle images. The other fundamental difference resides in the statistical evaluation of the particle displacement, which is recovered by performing a three-dimensional spatial correlation of particle locations. Since the three-dimensional location of particles is available, tracking is also a possible approach for lower density flows, which has allowed for various algorithms to be successfully implemented [54].

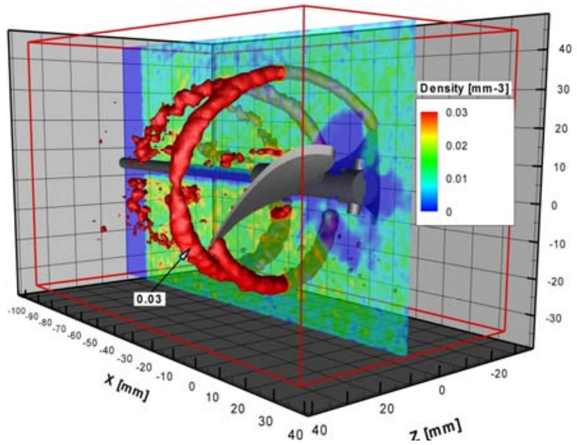

**Figure 3: Bubble density in a meridian plane and isosurface of density at 0.03 mm−3**

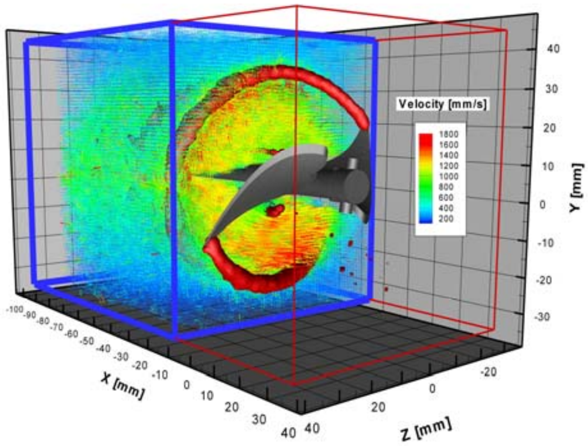

**Figure 4: Volumetric velocity field (partial) and isosurface as in Figure 3**

This technique has been applied mainly to two-phase flows, with the ultimate goal of providing the fluid velocity and the size characteristics of the bubble phase, e.g. population density and void fraction. For this reason, a method has been developed to retrieve the size properties of bubbles from their light scattering response [55]. The practice of defocusing DPIV has recently been applied to the two-phase flow-field around a propeller [56], the volume of which is 120 x 120 x 120 mm3. Figures 3 and 4 demonstrate the volumetric distribution of the bubble density and the volumetric three-dimensional velocity field, respectively.

### 4.4 Full Scale Measurements

Several methods are commonly used to measure the velocity distribution in the wake at both model and full-scales. While these measurements are routinely performed to determine the nominal wake for propulsor design and evaluation or for the validation of CFD codes, measurements of the velocity distribution in the wake at full-scale are relatively rare. For this reason, full-scale measurements were not developed.

There is some literature on the measurements of full-scale wake through the use of LDV and Pitot. Tanibayashi (1991) [57] used an LDV to measure the full-scale propeller inflow distribution as a basis of calculating propeller vibratory forces through the use of the training ship *Seiun Maru* as the full-scale ship. Kux et al. (1985) [40] used LDV systems to measure wake fields of model and full-scale ships. The ships used were the *Sydney Express* of Cb=0.61 Tanker model of and Cb=0.8. Three components of velocity are measured for the model ships, while one-component of velocity is measured for the full-scale ships. Sasajima's prediction method of full-scale wake estimation is validated through the comparison between predicted wake and measured wake patterns. Kuiper et al. (2002) [58] measured the wake in front of the working propeller equipped in a full-scale patrol boat (Lwl=36.5m) by using LDA. Norris (1984) [59] introduced an example of full-scale wake measurement through the use of LDV in the 17th ITTC.

Pitot tubes are also used to measure full-scale wake fields. Takahashi et al. (1971) [60] measured full-scale ship (Lpp=167m) wake distribution through the use of a Swing-arm type mount method. On the mount, five, five-hole Pitot probes and two, one-component Pitot probes were set with the measured plane 7.5 m ahead of A.P. The diameter of a five-hole Pitot is 60 mm and the apex angle is twenty degrees. Ogiwara shows the results of wake measurements for 3 different sized geosim models of a tanker ship (Lpp=7m, 30m and 300m) [61].

At DTNSRDC (1980), [62] rake-type Pitot-tubes were used to measure the wake at propeller plane for a fast patrol boat. Pressure measurements were carried out using an "air-blow system," similar to the method of Takahashi et al. (1971) [63]. A Pitot rake consisting of both a five and a thirteen hole Pitot tube was used. For the measurement of unsteady flow, five piezoelectric differential pressure transducers were mounted in the tube.

In 1981, Chai et al. [64] introduced an example of wake measurements for model and full-scale ships in which comb-type Pitot probes were used. The full-scale ship (Lpp=36.6m) was towed by a tug and the wake of 4 m long model ship was measured. The full-scale wake was predicted from the model ship wake through the use of Sasajima's method and compared with the measurement of a full-scale ship wake.

## 5. DEVELOPMENT OF PROCEDURES FOR MEASURING THE VELOCITY DISTRIBUTION IN THE SHIP WAKE AT MODEL SCALE

### 5.1 Introduction

The measurement of flow field by use of 5-hole Pitot tube is one of the common methods in towing tank testing. This method has a long history and it seems that almost all towing tank facilities have their own manual to use 5-hole Pitot to measure flow fields around ship. However, ITTC has not issued a procedure related to this method. It may be due to that it is too common method and no body may feel necessity to make a procedure. But since the measurement accuracy depends on the analysis method and it should be useful to issue the procedure in which calibration method, choice of probe and so on are documented.

### 5.2 Wake Measurements Procedures by 5-Hole Pitot

Procedure 7.5-02-03-02.5 describes the proper use of 5-Hole Pitot tubes to make wake measurements. Included in this procedure are guidance on:

- Instrumentation, model specs
- Calibrations
- Test procedure
- Data analysis
- Advantages and drawbacks of Pitot tubes

Included in the procedure are: the purpose for the procedure, analysis method, choice of probe, measurement method and uncertainty analysis. The choice of the size of Pitot probe, the effect of walls and the effect of shear flow are also mentioned. These procedures may also be useful in preparing and evaluating the results of the measurements.

For the uncertainly analysis, for instance, Olivieri et al. (2003) [65], Longo (2005) [66], Kume et al. (2006) [67] worked out the uncertainty analysis for the wake measurements. In these works, methods to analyze the uncertainty analysis for the wake measurements are introduced. However, the procedures seem not to be common and to introduce the procedure for uncertainty analysis, and only a brief introduction is described in the present procedure.

### 5.3 Wake Measurements Procedure by LDV

Nominal wake measurement is a typical test performed in the framework of ship design. The nominal wake data is required for propeller design. For these reasons the wake field committee has found it useful to issue a detailed procedure 7.5-02-03-02.4 to be used as a standard for the nominal wake measurement by LDV. In this procedure, detailed instructions are provided for any ITTC institution that seeks to perform this type of measurement. In particular, specifications are provided for instrumentation (traversing system, Laser Doppler Velocimetry system) and test models. The procedure also provides guidelines for calibration, test procedures and data analysis. Furthermore, the procedure can also be a useful guide to any flow survey, which uses LDV.

What you find in procedure 7.5-02-03-02.4 are guidelines for any ITTC institution on:

- Instrumentation
- Model specs
- Calibrations
- Test procedure
- Data analysis
- Advantages of LDV

## 6. REVIEW AND UPDATE OF THE EXISTING GUIDELINES FOR THE SIMULATION OF THE WAKE FIELDS FOR CAVITATION TESTING

By reviewing all of the ITTC Procedures for references to wake fields, it was found that only four existing procedures refer to the simulation of wake fields for cavitation testing. These procedures are:

- Procedure 7.5-02-03-03.1: Model-Scale Cavitation Test Cavitation Induced Pressure
- Procedure 7.5-02-03-03.3: Fluctuations Model Scale Experiments

- Procedure 7.5-02-03-03.5: Cavitation Induced Erosion on Propellers, Rudders and Appendages Model Scale Experiments
- Procedure 7.5-02-03-03.6: Podded Propulsor Model-Scale Cavitation Test

The committee's review of these procedures was made to ensure the accurate simulation of wake-fields for cavitation testing and to standardize the procedures for generating and assessing simulated wakes. Apart from standardizing the language, the main modifications introduced are:

- When a dummy model is used for testing, it is recommended that the simulated wake be verified by measurement of the wake and comparison to the actual wake.

- When wake measurements are performed the newly developed and submitted procedures (pitot tube and LDV) should be utilized.

The modifications to the above four procedures have been submitted to the Advisory Committee for adoption..

# 7. GUIDELINES FOR THE USE OF PIV IN DETERMINING THE VELOCITY DISTRIBUTION IN THE WAKE

## 7.1 Guidelines

The broad range of applications in which PIV can be used doesn't allow an easy step by step procedure to be provided. In the following sections some guidelines and practical information regarding PIV setups, which can help in the design of PIV experiments especially in towing tank, are given. The goal is not to provide detailed technical information on the measurement techniques that can be founded in more dedicated literature.

## 7.2 Laser

High energy pulsed laser are required to illuminate particles in PIV techniques and on the market are available Nd-Yag laser with a wide range of pulse energy up to 800 mJ at 532 nm and at repetition rate up to 30 Hz. For PIV applications in water the required energy density per area unit is in the range of 0.0005 to 0.001 mJ /mm$^2$ (for light sheet thickness of 1 mm and 10 µm hollow glass particles). Normally in towing tank testing, the area to be investigating is in the range from 100X100 mm$^2$ to 500X500 mm$^2$; normally a laser in the range of 100-200 mJ is enough for such application. For this range of energy compact lasers, with reduced dimension and weight of the head and power supplies, are available that allow easier handling on the towing tank carriage. An important parameter to be considered in the choice of the laser is the pulse repetition rate. In some lasers, the repetition rate cannot be changed or can be changed over a small range. In such a case the laser repetition rate should match exactly the camera frame rate or its multiples to assure the highest data rate of the PIV system.

## 7.3 Cameras

On the market are available a wide range of PIV cameras ranging from resolution of 1 million pixels (Mpx) to 12 Mpx. More information regarding camera specification and characteristics can be found in Raffel et al (2007) [68]. Higher resolution cameras normally have lower frame rate because of the PC bus throughput bottleneck. This is an important parameter to be taken into account in towing tank testing due the limited time in the run of the carriage. A compromise between the resolution and the frame rate shall be adopted. As a guideline, the optimum PIV resolution of the camera is considered to be in the range of 5 to 20 px /mm$^2$.

## 7.4 Seeding

Different type of seed material can be used for PIV testing and normally it is not an issue in PIV measurements in water. Seeding to water density ratios are usually almost always near unity, so consequently velocity lag can be normally neglected and it is also possible to use large particles up to 100μm in diameter. Large quantities of seeding material are required in towing tanks.  A good compromise between cost and effectiveness is provided by hollow glass particles that can be found easily in that they are commercially available with different dimensions. The choice of the seeding particle size shall be done considering the available laser energy. Normally 10 μm hollow glass particles require about $10^{-3}$ mJ/mm$^2$ laser energy, in the case of a low sensitivity camera. When lower energy is available larger particle dimensions or higher reflective particles (i.e. hollow glass particles silver coated) shall be used. The particles can be seeded in the flow by using a dedicated rake placed far upstream to avoid flow disturbance in the measurement area. To avoid such problems, in the towing tank testing, the rake can be placed downstream of the measurement area and the flow seeded in the return run of the carriage.

## 7.5 Stereo-PIV setup

Naval flows are strongly three-dimensional and the adoption of a stereo setup is an obvious option. A wide range of literature is available on the accuracy of SPIV and the advantage and drawbacks of the different optical configuration [68].  The symmetrical configuration, with the cameras placed symmetrically on both side of the light sheet in forward scatter is the most performing from the accuracy point of view. However is not practical due to the limited optical access in the wake of ship model. Therefore the optical setup with both cameras placed from the same side of the light sheet allows an easier optical access. However care shall be taken in order to assure an angle between the cameras is larger than 20° to assure enough accuracy in the stereo reconstruction.

### 7.5.1 Underwater SPIV probes

In towing tank application underwater cameras and light sheet are required. Underwater SPIV probes have been developed for application in towing tank testing [68]. In the choice of such systems, flexibility and capabilities in changing the SPIV optical configuration shall be of primary importance to adapt the optical configuration to the different needs in the towing tank testing.

### 7.5.2 SPIV Calibration

The calibration of a PIV system is a crucial point for the quality of the final result of the measurement especially for Stereo PIV system. The angular setup and the Solof method (Solof et al, 1997 [69]) are widely used. Calibration can be performed using commercially available three-dimensional calibration targets, which allow for a simpler calibration procedure. However a more accurate calibration result is obtained by displacing a two dimensional calibration target over several planes. During calibration, the alignment of the target with the light sheet is crucial and impacts the accuracy of the reconstruction. In most of the commercially available SPIV software, an algorithm is implemented that evaluates the misalignment of the light sheet with the calibration target and corrects the error from the PIV images. Normally, if the misalignment is small (mismatch of the angles between light sheet and target plane within 1 degree) the correction works well. For larger misalignment errors, it is better to recalibrate the SPIV system trying to better the alignment between the light sheet and the target.

If possible, after calibration is complete it is recommended that a test of the calibration be performed, e.g. measuring a well known uniform flow, which can be easily obtained in a towing tank.  A good calibration shows differences between the true and measured velocity within 2% over 95% of the measurement area. Larger errors are normally located at the corners of the measurement area.

### 7.6 PIV timing

The timing of the laser pulse is of primary importance in the quality of the measurement. Pulse separation time shall assure particle displacements of the order of at least 5 pixels to reduce as much as possible the weight of the sub-pixel interpolation in the final result. In cross-flow measurements, when the main flow is orthogonal to the light sheet, the time separation can be limited by the light sheet thickness because of the loss of particles leaving the light sheet that are dropping the correlation peak. In such case the laser optics shall provide a thicker light sheet to assure the above minimum displacement.

### 7.7 Data Management and Processing.

The primary raw data of the PIV measurement are the images. For an instantaneous stereo PIV measurement, with 4 Mpx -12 bit/px cameras, the size of the data is 32 Mb. It's very easy in a test campaign to obtain hundreds of Gigabytes to be analyzed. Normally saving, restoring and data analysis is a time consuming task. A rough rule of thumb is that with the available computer technology (3 Ghz multicore- multiprocessor PC) and the available state of the art PIV algorithms the data acquired in 1 day of towing tank requires at least 5 days for processing. A lot of effort has been spent in the last decade in improving the image processing algorithms. The results of the PIVCHALLENGE 2003 [70] and 2005 [71] have shown that windows deformations algorithms provide a higher accuracy in the particle displacement evaluation. However the improvement in accuracy is obtained with an increased computational effort and processing large data sets is a time consuming task. Normally when evaluating mean flow over many instantaneous velocity fields, a faster but less accurate algorithm provides acceptable results.

### 7.8 Reflections

Light reflections coming from model surface or from bright objects in the background modify the grey levels in the PIV images and affect the measurement accuracy. There are no image algorithms able to remove such effects and the only possible approach is to reduce as much as possible the problem, painting in black any object in the field of view of the PIV cameras. For small setups, when the volume of the flow to be seeded is small, it is possible to use very costly fluorescent particles which are reemitting the incident light at a different frequency. In such way is possible to separate the effect of reflection from particle image by using a dedicated interferential filter in front of the camera objective.

## 8. RECOMMENDATIONS TO THE CONFERENCE

The 25th ITTC Specialist Committee on Wake Fields recommends the adoption of the changes to the four existing procedures:

- Procedure 7.5-02-03-03.1: Model-Scale Cavitation Test Cavitation Induced Pressure
- Procedure 7.5-02-03-03.3: Fluctuations Model Scale Experiments
- Procedure 7.5-02-03-03.5: Cavitation Induced Erosion on Propellers, Rudders and Appendages Model Scale Experiments
- Procedure 7.5-02-03-03.6: Podded Propulsor Model-Scale Cavitation Test

and the acceptance of the two new procedures on the use of 5-Hole Pitot Tubes (Procedure 7.5-02-03-02.5) and LDV (Procedure 7.5-02-03-02.4) to measure nominal wakes at the propeller plane.

Additionally, the committee recommends that the ITTC:

- Monitor and review the development of PIV as a more standard measurement system
- Survey and monitor validation of numerical predictions of wake fields at full-scale.

Both these areas continue to become more common.

## 9. REFERENCES AND NOMENCLATURE

### 9.1 References

[1] CFD WORKSHOP TOKYO, 2005, National Maritime Research Institute, Japan.

[2] Rhee S.H., Fred Stern F., 2001,"Unsteady RANS method for surface ship boundary layer and wake and wave field", International Journal For Numerical Methods In Fluids, vol 37: pp.445–478

[3] Alin N., Fureby C., Svennberg U., 2003, "LES of the Flow Past Simplified Submarine Hulls," 8th International Conference on Numerical Ship Hydrodynamics.

[4] Bensow R.E., Fureby C., Liefvendahl M., Persson T., 2006, "A Comparative Study of RANS, DES and LES," 26th Symposium on Naval Hydrodynamics, p.13.

[5] Bugalski T., Kraskowski M., 2006, "Validation of the RANSE wake computations for the training vessel Nawigator XXI and the dredger Uilenspiegel**",** 9th Numerical Towing Tank Symposium, France

[6] Hino T., 2006, "CFD-Based Estimation of Propulsive Performance in Ship Design," 26th Symposium on Naval Hydrodynamics, p.12.

[7] Kim J., Park I.-R., Van S.-H., Choi Y.-B., Park N.-J., 2006, "Numerical Computation for the Comparison of Stern Flows around Various Twin Skegs," Journal of Ship & Ocean Technology SOTECH, Vol.10, No., 2, pp.1-10.

[8] Kimura K., Taketani T., Yamasaki E., Fujii, A., 2007, "Study on Interaction between Ship Hull and Propeller using RANS Method with VLM," pp. 655-663.

[9] Hino T., Ohashi K., Ukon Y., 2003, "Flow Computations around a Ship with Appendages by an Unstructured Grid Based NS Solver," 8th International Conference on Numerical Ship Hydrodynamics.

[10] Laurens J-M., Cordier S., 2003, "Numerical Simulation of Propeller-Rudder Interface," 8th International Conference on Numerical Ship Hydrodynamics.

[11] Regnstrom, B., Bathfield, N., 2006, "Drag and Wake Prediction for Ships with Appendages Using an Overlapping Grid Method," 26th Symposium on Naval Hydrodynamics, p.16.

[12] Bensow R.E., Liefvendahl M., Wikstorm N., 2006, "Propeller Near Wake Analysis Using LES with a Rotating Mesh," 26th Symposium on Naval Hydrodynamics, p.9.

[13] Tahara et al, 2003, "Computation of Ship Viscous Flow at Full Scale Reynolds Number," Journal of the Society of Naval Architects of Japan, Vol. 192.

[14] Schweighofer J., 2004, "Numerical Investigation of the Turbulent Free-

Surface Flow around the Series 60 Ship at Model- and Full-Scale Ship Reynolds Numbers", Yearbook of the Society of Naval Architects of Germany, STG Summer Meeting in Poland

[15] Visonneau M., 2005, "A step towards the numerical simulation of viscous flows around ships at full scale - recent achievements within the European union project EFFORT", Marine CFD Conference, Southampton, UK

[16] Visonneau, M., Deng G.B., Queutey P., 2006, "Computation of Model and Full Scale Flows around Fully-Appended Ships with an Unstructured RANSE Solver," 26th Symposium on Naval Hydrodynamics, p.16.

[17] Schweighofer J., Regnstrom B., Starke A.R., Tzabiras G., 2005, "Viscous-flow computations of two existing vessels at model- and full-scale ship Reynolds numbers - a study carried out within the European union project, EFFORT", International Conference on Computational Methods in Marine Engineering – MARINE, Oslo, Norway

[18] Starke, B., Windt, J., Raven, H., 2006, "Validation of Viscous Flow and Wake Field Predictions for Ships at Full Scale," 26th Symposium on Naval Hydrodynamics, p.18.

[19] Verkuyl J.B., Visonneau M., Hänninen S., de Jager A., 2006, "Validated Full Scale Flow Computations Speed up the European Design Process, A Practical Overview of the Results of the European Union Project, EFFORT," SNAME

[20] Han K-J., Larsson L., Regnstrom B., 2006, "Numerical Optimization of the Propeller Behind a Ship Hull at Full Scale," 26th Symposium on Naval Hydrodynamics, p.12

[21] Pien, P.C 1958, "Five-Hole Spherical Pitot Tube," David Taylor Model Basin Hydromechanics Laboratory, Research and Development Report, Report No.1229

[22] Wright, M.A., 1970, "The Evaluation of a Simplified Form of Presentation for Five-Hole Spherical and Hemispherical Pitometer Calibration data," Journal of Physics E: Scientific Instruments Vol. 3

[23] Fujita, 1979, "On the Flow Measurement in High Wake Region at the Propeller Plane," Journal of the Society of Naval Architects of Japan, Vol.145, pp.1-7

[24] Olivieri, A., 2003, "Experimental Investigation of the Flow Around a Fast Displacement Ship Hull Model, Journal of Ship Research, Vol. 47, No.3, Sept., pp.247-261

[25] Felli, M. and Di Felice, F., 2005, "Propeller Wake Analysis in Non Uniform Inflow by LDV Phase Sampling Techniques," Journal of Marine Science and Technology, Vol. 10.

[26] Felli, M. and Di Felice, F., 2002, "Ship Model Wake Visualization by LDV in a Towing Tank," 10th International Symposium on Flow Visualization, Kyoto, Japan.

[27] Felli, M. and Di Felice F., 2004, "Analysis of the Propeller-Hull Interaction by LDV Phase Sampling Techniques," Journal of Visualization, Vol. 7, N.1.

[28] Felli, M. and Greco, L., Colombo, C., Salvatore, F., Di Felice, F., Soave, M., 2006, "Experimental and Theoretical Investigations of Propeller-Rudder Interaction Phenomena," 26th Symposium on Naval Hydrodynamics, Rome, Italy.

[29] Micheal, T.J. and Chesnakas, C.J., 2004, "Advanced Design, Analysis and Testing of Waterjet Pumps," 25th Symposium on Naval Hydrodynamics, St John's, Newfoundland and Labrador, Canada.

[30] Jessup, S., Chesnakas, C.J., Fry, D., Donnelly, M., Black, S., Park, J., 2004, "Propeller Performance at Extreme Off Design Conditions," 25th Symposium on Naval Hydrodynamics, St John's, Newfoundland and Labrador, Canada.

[31] Felli M., Di Felice, F., Lugni, C., 2004, "Experimental Study of the Flow Field Around a Rolling Ship Model," 25th Symposium on Naval Hydrodynamics, St John's, Newfoundland and Labrador, Canada.

[32] Raffel, M., Willert, C., Kompenhans, J., 1998, "Particle Image Velocimetry," Springer ISBN 3-540-63683-8.

[33] Dong, R., Katz, J., Huang, T., 1997, "On the Structure of Bow Waves on a Ship Model," Journal of Fluid Mechanics, Vol. 346, pp. 77-115.

[34] Cotroni, A., Di Felice, F., Romano, G. P., Elefante, M., 2000, "Investigation of the Near Wake of a Propeller Using Particle Image Velocimetry," Experiments in Fluids 29, S227–S236 suppl.

[35] Di Felice, F., Di Florio, D., Felli, M., Romano, G.P., 2004, "Experimental Investigation of the Propeller at Different Loading Conditions Wake by Particle Image Velocimetry," Journal of Ship Research, Vol. 48, No. 2, pp. 168–190.

[36] Felli, M., Di Felice F., Gui, G., Camusi R., 2006, "Analysis of the Propeller Wake Evolution by Pressure and Velocity Phase Measurements,'' Experiments in Fluids, Vol. 41, pp. 441–451.

[37] Paik, B.-G., Kim, J., Park, Y.-H., Kim, K.-S., Yu, K.-K., 2007, "Analysis of Wake Behind a Rotating Propeller Using PIV Technique in a Cavitation Tunnel," Journal of Ocean Engineering, Vol. 34, Issues 3-4, pp. 594-604.

[38] Calcagno, G., Di Felice, F., Felli, M., Pereira, F., 2002, "Propeller Wake Analysis Behind a Ship by Stereo PIV," Proceedings of the 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan..

[39] Jessup, S., 2004, "Unsteady Propeller Performance in Crashback Conditions with and without a Duct," 25th Symposium on Naval Hydrodynamics, St. John's, Canada.

[40] Jessup, S., Fry, D., Donnelly, M., 2006, "Unsteady Propeller Performance in Crashback Conditions with and without a Duct," 26th Symposium on Naval Hydrodynamics, Rome, Italy.

[41] Gui, L., Longo, J., Stern, F., 2001, "Towing Tank PIV Measurement System, Data and Uncertainty Assessment for DTMB Model 5512," Experiments in Fluids, Vol. 31, pp. 336–346.

[42] Felli, M., Pereira, F., Calcagno, G., Di Felice, F., 2003, "A Modular Stereo-PIV Probe for Underwater Applications: Configurations and Measurement Performance," PIV03, Busan, Korea.

[43] Di Felice F., Pereira F., "Developments and Application of PIV in Naval Hydrodynamics" Particle Image Velocimetry New Developments and Recent Applications Series: Topics in Applied Physics, vol 112, A .Schroder, C.E.Willert (Eds.), pp. 459–485 (2007), Springer-Verlag

[44] Abe, M., Yoshida, N., Hishida, K., Maeda, M., 1998, "Multilayer PIV Technique with High Power Pulse Laser Diodes," 9th International Symposium, Applied Laser Technology Fluid

Mechanics, Instituto Superior Tecnico, Lisbon, Portugal.

[45] Brücker, C., 1997, "3D Scanning PIV Applied to an Air Flow in a Motored Engine Using Digital High-Speed Video," Measurement Science and Technology, Vol. 8, pp. 1480–1492.

[46] Kähler, C.J., Kompenhans, J., 2000, "Fundamentals of Multiple Plane Stereo Particle Image Velocimetry," Experiments in Fluids, Vol. 29, pp. 070–S077.

[47] Meng, H., Pan, G., Pu, Y., Woodward, S.C., 2004, "Holographic Particle Image Velocimetry: From Film to Digital Recording" Measurement Science and Technology, Vol. 15, pp. 673–685.

[48] Malik N. A., and Dracos, T., Lagrangian, J., 1993, "PTV in 3D Flows," Journal of Applied Science and Research, Vol. 51, pp. 161–6.

[49] Elsinga, G.E., Scarano, F., Wieneke, B., Van Oudheusden, B.W., 2006, "Tomographic Particle Image Velocimetry," Experiments in Fluids, Vol. 41, pp. 933-947.

[50] Pereira, F., Gharib, M., Dabiri, D., and Modarress, D., 2000, "Defocusing Digital Particle Image Velocimetry: A 3-component 3-dimensional DPIV Measurement Technique, Application to Bubbly Flows," Experiments in Fluids, Vol. 29, pp. 78–84.

[51] Katz, J., Gowing, S., O'Hern, T., and Acosta, A., 1983, "A Comparative Study Between Holographic and Light-Scattering Techniques of Microbubble Detection," Proc. Symposium, Measuring Techniques in Gas-Liquid Two-Phase Flows, Nancy, France, IUTAM, pp 41–66.

[52] Malkiel, E., Alquaddoomi, O., and Katz, J., 1999, "Measurements of Plankton Distribution in the Ocean Using Submersible Holography" Measurement in Science and Technology, Vol. 10, pp. 1142–52.

[53] Pereira, F., Gharib, M., 2002, "Defocusing Digital Particle Image Velocimetry and the Three-Dimensional Characterization of Two-Phase Flows," 2002, Measurement in Science and Technology, Vol. 13, pp. 683–694.

[54] Pereira, F., Stüer, H., Castaño-Graff, E., Gharib, M., 2006, "Two-Frame 3D Particle Tracking," Measurement in Science and Technology, Vol. 17, pp. 1680–1692.

[55] Pereira, F., Gharib, M., 2004, "A Method for Three-Dimensional Particle Sizing in Two-Phase Flows," Measurement in Science and Technology, Vol. 15, pp. 2029–2038.

[56] Pereira, F., Castaño-Graff, E., Gharib, M., 2006, "Bubble Interaction with a Propeller Flow," 26th Symposium on Naval Hydrodynamics, Vol. 1, Rome, Italy, pp. 299–308.

[57] Tanibayashi et al., 1991, "Full-Scale LDV Measurements of Ship Stern Flow", Osaka Colloquium.

[58] Kux et al., 1985, "Correlation of Wake Measurements at Model and Full Scale Ship," 15th Symposium on Naval Hydrodynamics, Washington, D.C.

[59] Kuiper et al, 2002, "Propeller Inflow at Full Scale During a Manoeuvre," 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.

[60] Norris C., 1984, "On the Use of Laser Velocimetry in the Full-Scale Ship's Environment," Proceedings of the 17th ITTC.

[61] Takahashi, 1971, "Measurement of Velocity Distribution ahead the Propeller

Disk of the Ship," Transactions of the West-Japan Society of Naval Architects No.42. pp. 153-168.

[62] Ogiwara S., 1994, "Stern Flow Measurements for The Tanker "Ryuko-Maru" in Model Scale, Intermediate Scale and Full Scale Ships," Proceedings of CFD Workshop Tokyo, Vol.1, pp.341-349

[63] Day, W. et al., 1980, "Full-Scale Propeller Disk Wake Survey and Boundary layer Velocity Profile Measurements on the 154-Foot Ship R/V Athena," DTNSRDC report 0833-01.

[64] Chai et al., 1981 "Comparison and Analysis of the Wake Distribution of the Passenger Ship *Hong Xing 514* and its Model," Proceedings of the 16th ITTC.

[65] Olivieri, A., Pistani, F., 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.

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

[67] Kume K., Hasegawa J., Tsukada Y., Fujisawa J., Fukasawa R., Hinatsu M., 2006, "Measurements of Hydrodynamic Forces, Surface Pressure, and Wake for Obliquely Towed Tanker Model and Uncertainty Analysis for CFD Validation," Journal of Marine Science and Technology, Vol.11, pp.65-75.

[68] Raffel M., Willert C., Wereley S., Kompenhans J., 2007, "Particle Image Velocimetry Springer Verlag, 2nd edition, ISBN: 978-3-540-72307-325th

[69] Solof SM, Adrian RJ, Liu ZC, 1997, "Distortion Compensation for Generalized Stereoscopic Particle Image Velocimetry," Measurement in Science and Technology, Vol. 8, pp. 1441–1454

[70] Stanislas, M., Okamoto, K., and Kähler, C., 2003, "Main results of the First International PIV Challenge," Meas. Sci. Technol. Vol. 14, pp. R63-R89

[71] Stanislas, M., Okamoto, K., Kähler, C., and Westerweel, J., 2005, "Main results of the Second International PIV Challenge" Vol. 39, Number 2, August, pp.170-191

## 10. LITERATURE SURVEY

### 10.1 Numerical Methods for Wake-Field Prediction

1. Berchiche, N., Janson C-E., 2005, "Flow Simulation of a Propeller in Open Water Condition and Experimental Validation," NuTTS.

2. Bugalski T., 2007, "An overview of the selected results of the European Union Project EFFORT", Hydronav07, Archives of Civil and Mechanical Engineering Polish Academy of Sciences,Vol.7, No.3, ISSN 1644-9665

3. Bugalski T., 2007, "Study on numerical prediction of effective wake field", 10th Numerical Towing Tank Symposium (NuTTS), Hamburg, Germany

4. Carrica P.M., Wilson R.V., Stern F., 2006, "Unsteady RANS Simulation of the Ship Forward Speed Diffraction Problem," Computers & Fluids, Vol. 35, pp. 545–570.

5. Choi J-E. , Min K-S., Chung S-H., Seo H-W., 2003, "Study on the Scale Effects on the Flow Characteristics around a Full Slow-Speed Ship," 8th International

Conference on Numerical Ship Hydrodynamics.

6. Ebert M.P, Gorski J., Coleman R.M, 2003, "Viscous Flow Calculations of Waterjet Propelled Ships," 8th International Conference on Numerical Ship Hydrodynamics.

7. Greco L., Salvatore F., Di Felice F., 2004, "Validation of a Quasi–Potential Flow Model for the Analysis of Marine Propellers Wake," 25th Symposium on Naval Hydrodynamics, p.21.

8. Hyman M.C., Moraga F.J., Drew D.A., Lahey R. T. Jr., 2006, "Computation of the Unsteady Two-Phase Flow around a Maneuvering Surface Ship," 26th Symposium on Naval Hydrodynamics, p.20.

9. Jacquin E., Guillerm, P.-E., Drouet, A., Perdon, P., 2006, "Simulation of Unsteady Ship Maneuvering Using Free-Surface RANS Solver," 26th Symposium on Naval Hydrodynamics, p.19.

10. Kinnas, S.A., Mohammed, F., Lee, H., 2006, "Propeller Tip Vortex/Rudder Interaction," 26th Symposium on Naval Hydrodynamics, p.15.

11. Lubke L.O., Mach K.-P., 2004, "LDV Measurements in the Wake of the Propelled KCS Model and its use to Validate CFD Calculations," 25th Symposium on Naval Hydrodynamics, p. 10.

12. Luquet R., Gentaz, L., Ferrant, P., and Alessandrini, B., 2004, "Viscous Flow Simulation Past a Ship in Waves Using the SWENSE Approach," 25th Symposium on Naval Hydrodynamics, p.17

13. Maki K.J., Doctors L.J., Beck R.F., 2007, "On the Profile of the Flow Behind a Transom Stern," 9th International Conference on Numerical Ship Hydrodynamics, Ann Arbor, Michigan.

14. Mascio A. (Di), Broglia R., Muscari R., 2007, "Numerical Simulation of Viscous Flow around a Fully Appended Hull with Enforced Motion," 9th International Conference on Numerical Ship Hydrodynamics, Ann Arbor, Michigan.

15. Miller, R., Gorski, J., 2006, "Resistance Predictions of High Speed Mono and Multihull Ships with and without Water Jet Propulsors using URANS," 26th Symposium on Naval Hydrodynamics, p.16.

16. Rhee B., Sung C-H., Koh I-Y., 2003, "Validation of the Flow around an Appended Suboff Body Using Parallelization and a New Wall Method," 8th International Conference on Numerical Ship Hydrodynamics.

17. Starke B., Ploeg van der A., 2007, "Computation of Transom-Stern Flows Using a Steady Free-Surface Fitting RANS Method," 9th International Conference on Numerical Ship Hydrodynamics, Ann Arbor, Michigan.

18. Van S-H., Kim J., Park I-R., Kim W-J., 2003, "Calculation of Turbulent Flows around a Submarine for the Prediction of Hydrodynamic Performance," 8th International Conference on Numerical Ship Hydrodynamics.

19. Vysohlid, M., Mahesh, K., 2006, "Large Eddy Simulation of Crashback in Marine Propellers," 26th Symposium on Naval Hydrodynamics, p.11.

20. Yavuz I., Cehreli Z.N., Celik I.B., 2004, "Comparison of Detailed Simulations of a Turbulent Ship Wake on a Straight and Circular Track," 25th Symposium on Naval Hydrodynamics, p.13.

21. Yu H-R., Park W-G., 2003, “Numerical Flow Analysis of Ducted Propeller,” 8th International Conference on Numerical Ship Hydrodynamics.

22. Zhao F., Zhang Z.R., Chang Y., 2006, “Practical Application of CFD in Wake Simulation of Hull Model with Various Appendages,” 7th International Conference on Hydrodynamics, pp. 641-6467

## 10.2 Experimental Methods

1. Bu-Geun, P., Kim, J., Park, Y., Kim, K., Yu, K., 2005, “Measurements of Flow around a Rotating Propeller Using PIV Technique in a Cavitation Tunnel,” 6th International Symposium on Particle Image Velocimetry, Pasadena, CA.

2. Di Felice, F., Dolcini, A., Pereira, F., Lugni, C., 2005, “Flow Survey around the Bilge Keel of a Ship Model in Free Roll Decay,” 6th International Symposium on Particle Image Velocimetry, California Institute of Technology, Pasadena, CA.

3. Schröder, A., Willert, C., (Eds.) “Particle Image Velocimetry New Developments and Recent Applications Series: Topics in Applied Physics,” Vol. 112.