Specialist Committee on CFD in Marine Hydrodynamics

Final report and Recommendations to the 27th ITTC

1. INTRODUCTION

1.1 Membership

Chairman:
Prof. Takanori Hino
Yokohama National University, JAPAN

Secretary:
Prof. Pablo Carrica
The University of Iowa, USA

Members:
Dr. Riccardo Broglia
Istituto Nazionale per Studi ed Esperienze di Architettura Navale (INSEAN), ITALY

Mr. Peter Bull
QinetiQ, UK

Dr. Sung-Eun Kim
Naval Surface Warfare Center, Carderock Division, USA

Dr. Da-Qing Li
SSPA Sweden AB, SWEDEN

Prof. Decheng Wan
Shanghai Jiao Tong University, CHINA

Prof. Shin Hyung Rhee
Seoul National University, SOUTH KOREA

Mr. Ilkka Saisto
VTT, Ship Hydrodynamics, FINLAND

Dr. Ignazio Maria Viola
The University of Edinburgh, UK

1.2 Meetings

The committee met 4 times:
26-27 March 2012, Yokohama, Japan
8-9 Nov 2012, Bethesda, MD, USA
10-11 June 2013, Jeju, South Korea
16-17 January 2014, Portsmouth, UK

1.3 Tasks

The purpose of this specialist committee is to comprehensively review past work on areas treated separately by previous committees. General conclusions on the status of practical applications of CFD and suggestions for future CFD applications will be beneficial to all members of ITTC.

1. Review, from an interdisciplinary perspective, the current status of CFD in areas of importance to ITTC. Include resistance, propulsion, propulsors, manoeuvring, steep and breaking wave simulation, seakeeping, ocean engineering and steady and unsteady flow field prediction at model and full scale.
2. Review developments and identify need for research in steady and unsteady CFD at full

scale, including real-time CFD for the use in manoeuvring simulators.

3. Identify benchmark data needed for validation. List requirements for experiments. Create a list of benchmark experiments for validation of different aspects of CFD for ship hydrodynamics and offshore structures, including output needed from the experiments and the level of uncertainty required.

4. Check the need for formal procedures and guidelines on CFD verification and validation in specific areas.

5. Update the guidelines 7.5-03-02-03, Practical Guidelines for Ship CFD Applications.

6. Review use and validation of CFD for wake scaling and determination of nominal full-scale wakes.

7. Develop procedures for RANS simulation of model and full scale nominal wakes.

8. Review recent developments in techniques for direct numerical simulation of wakes.

2. STEEP AND BREAKING WAVES

Steep and breaking waves are of interest in the context of producing a seaway for seakeeping stability and manoeuvring simulation, to study impact loads of waves against objects, and to analyze bubble entrainment and aeration.

Simulation of steep and breaking waves requires advanced CFD tools to accurately resolve the air/water interface. Turbulence production and dissipation, bubble entrainment and breakup/coalescence and capillary wave formation all play a role on the dynamics of a breaking wave. Classical methods, like boundary integral equation and high-order spectral, fail when waves get close to overturning and ignore the presence of air (Brucker et al. 2010). Immersed interface methods like level set (LS) and volume of fluid (VOF) are mostly used to resolve complex topologies resulting from steep and breaking waves.

The scales involved in breaking waves around ships are vast and direct numerical simulation of all the processes involved in breaking waves and consequent air entrainment will be out of reach for the foreseeable future. Considering a ship of length L and a computational domain of size 3 L^3 where bubbles down to 50 μm in diameter (a size commonly present in full scale ships, Johansen et al. 2010) are resolved with 10 grid points. For L = 100 m a uniform grid would require 2.4×10^{22} points, still too coarse to properly resolve bubble coalescence where liquid film thinning before coalescence can be less than 1 μm in thickness. Using 2.4 million points per core, still 10,000 trillion cores would be required for such computation, orders of magnitude more than all the cores available in all computers ever manufactured on earth combined.

Air entrainment produced by breaking waves is then simulated using smaller scale problems and Eulerian approaches with air entrainment models (Carrica et al. 2012a), where the breaking wave is resolved using a Detached Eddy Simulation (DES) approach but the bubbles are modelled as a transported field. A good dataset is available from the experiments of Tavakolinejad et al. (2010) on a 2D+T breaking wave resembling the bow wave of DTMB 5415.

Direct simulation of breaking waves ignoring the scales required to fully resolve bubbles are within the possibilities of current computational capabilities. In a pioneering work, Lubin et al. (2006) studied a plunging breaker using a Large-Eddy Simulation (LES) approach. Later Lubin and Glockner (2013) simulated a breaking wave using similar methodologies with a finer grid with 83 million cells, analysing the vertical structures produced during the breaking process. Brucker et al. (2010) performed simulations of a three dimensional breaking wave using VOF and an implicit sub-grid scale stress model on 3 Cartesian grids with up to 134 million points. A pressure forcing technique was developed that allows generation of fully nonlinear progressive waves in a periodic domain. The authors analysed the different stages of the breaking process and computed detailed balances of energy and air entrainment.

Massive simulations of a wedge-induced breaking wave were performed by Wang et al. (2012) on 2 billion grid points. The authors used a sharp-interface method to resolve the interface and LES as turbulence model. Results were compared with experimental data with good agreement. Analysis also included bubble entrainment and filament breakup.

Extreme waves can be generated in a towing tank by focusing linear waves such that they coincide at a designed time and location, forming constructive interference. Mousaviraad et al. (2008) used a similar technique in CFD to produce a "three sisters" wave event and run an auto-piloted ONR Tumblehome combatant through it. The results show that survivability studies can be performed using this methodology to create transient extreme events at moderate computational cost.

Of primary interest for seakeeping, manoeuvring and stability simulations is the generation of a realistic seaway. Standard CFD simulation of ships in waves requires initial and boundary conditions to start and propagate nonlinear waves inside the computational domain. Analytical solutions exist to impose linear waves and linear superposition produces acceptable oceanic waves for many applications, with JONSWAP, Pierson-Moskowitz or Bretschneider spectra typically used (Mousaviraad et al. 2008). However, large-amplitude waves are nonlinear and, though they deform and evolve significantly within the computational domain, the domain size is typically too small and the resolution too coarse to develop a realistic seaway. Steep waves are nonlinear and no analytical solutions exist that can be used as initial and boundary conditions, and at the same time spectra cannot be easily generated since linear superposition of nonlinear waves is not valid.

Direct simulations of nonlinear ocean waves have been performed by Xiao et al. (2013) using the high-order spectral (HOS) method, and by Dommermuth et al. (2013) and Rottman et al. (2013) using VOF. While HOS enables prediction of large-amplitude rogue waves, it cannot model wave breaking. Dommermuth et al. (2013) use a process to assimilate data from HOS computations of JOSWAP or Pierson-Moskowitz into VOF simulations, enabling investigation of turbulent flows and breaking waves that are not possible using HOS or field measurements. In the case of Rottman et al. (2013) computations and posterior developments by the authors, extremely fine grids of up to 17.2 billion points were used on 8192 processors on a Cray XE6 supercomputer. The simulations were performed for a wind speed of 11.1 m/s and a peak wavelength of the spectrum of 100 m. The resulting seaway possesses the most detailed physics available to date, see Fig. 2.1. A section of the seaway can then be used as initial and transient boundary conditions for CFD computations including a ship to study seakeeping or stability on a realistic seaway.

In summary, steep and breaking wave simulations are feasible with today's computational resources and CFD techniques to the scale of a few wavelengths and as long as direct simulation of bubble entrainment is not required. Direct simulation of bubble entrainment is typically limited to small domains.

Figure 2.1. Spilling waves on a seaway simulated with VOF (Rottman et al. 2013)

3. STEADY AND UNSTEADY FLOW FIELD PREDICTIONS

Most flow fields appearing in practical marine hydrodynamics are unsteady in nature. Due to the high-Reynolds numbers involved in ship flows, both in model and full scale, unsteadiness will always be present due to turbulent fluctuations. However, steady flows in the Reynolds averaged sense are possible, and we refer to unsteadiness when the averaged flow field is unsteady. One steady flow field occurs for the resistance prediction of a ship running straight ahead with a constant speed in calm water. Steady flow fields also appear in other situations such as the steady drift and the steady turning in manoeuvring applications. Self-propulsion simulations are time-dependent due to propeller rotation. However, the body force propeller models often used in the simplified hull-propeller interaction analysis are usually incorporated with time-averaged flow fields and therefore steady flow approaches can be applied. In other applications of ship hydrodynamics, flow fields are inherently unsteady.

Unsteadiness comes from ship motions, ambient waves, or motions of appendages and propellers. In ocean engineering applications, the situation is similar to ship applications and the ambient waves and the motions associated with them or the vortex shedding and the vortex induced motions are typical sources of unsteadiness.

In the past, CFD applications were focused on resistance predictions of ships, since it is the simplest of flow fields. In terms of numerical procedures, some algorithms are designed to obtain efficiently a steady state solution. The artificial compressibility approach is one of such approaches, in which a flow field does not satisfy mass conservation until steady state is achieved. The SIMPLE algorithm is also used for steady flow simulations in which time marching is not used and an iterative procedure is adopted instead of coupling the velocity and pressure fields to enforce mass conservation. On the other hand, for unsteady flow approaches the mass conservation must be satisfied at each time step, which requires an iteration process with the pressure Poisson equation in order to cope with the elliptic nature of pressure fields and the nonlinearity of the coupled system of equations. As a result, unsteady computations require longer CPU times than steady state computations.

Another issue in unsteady approaches is the discretization in time. As in the case of spatial discretization, the order of accuracy and the time step size affect the quality of the solutions. It is well known that first order time integration schemes introduce significant errors both for temporal and spatial accuracy in unsteady computations. Higher-order schemes are thus recommended for time discretization. Sometimes a steady state solution is computed using unsteady flow codes, and in such cases first order time schemes can be used safely since the final steady state solutions do not depend on the temporal schemes. Time step size must be, first of all, determined by stability considerations. Each combination of temporal and spatial discretization has its own stability limit and the time step size is restricted by it. Usually the implicit schemes such as the Euler backward method or the three-step backward scheme allow larger time steps compared with explicit schemes. Although implicit schemes need additional numerical operations and longer CPU time per time step than explicit ones, the gain from larger time steps is so prominent that implicit schemes are preferred in practical applications.

Time step size must also be determined by the requirements from the flow physics. For example, if regular incident waves are present, the time step size is set in accordance with the resolution needed for the wave period. Similar considerations are applied in manoeuvring motions, propeller rotations, or any other physical problem that is being solved.

Practical applications of steady and unsteady flow predictions are reviewed in two recently held CFD workshops, SIMMAN 2008 Workshop (Stern et al. 2011) for manoeuvring simulations and Gothenburg 2010 Workshop on CFD for Ship Hydrodynamics (Larsson et al. 2014). Although both workshops include test cases for steady and unsteady flow fields, the questionnaires regarding numerical methods are organized in solver by solver base and give a wide view of CFD codes in ship hydrodynamics.

Approximately 80% of codes participating in the workshops are designed for unsteady flow problems, although the majority of the test cases require steady state solutions. Unsteady flow solvers seem to be the workhorse in marine hydrodynamics applications.

Velocity-pressure coupling in most solvers is accomplished by pressure correction methods and their variants. The direct method in which the momentum equations and the continuity equation are directly coupled is used by some codes. For steady state solvers, SIMPLE method or artificial compressibility are typically used for coupling pressure and velocity

The most used time discretization scheme is the first-order Euler implicit scheme. In cases where steady flow solutions are computed, the Euler implicit scheme is the natural choice for the unsteady solvers since time accuracy is not needed and a large time step is desirable for faster convergence. The second favored method is the three-step backward scheme, also referred as the second-order implicit scheme. For the unsteady flow problems, this choice of time discretization scheme seems a good compromise between the accuracy and the numerical complexity. Other schemes, (Euler explicit, Runge-Kutta or Adams-Bashforth) are used by a limited number of codes.

In summary, for ship hydrodynamics applications, unsteady flow solvers are most frequently adopted with pressure correction method or direct method for velocity-pressure coupling. Time discretization schemes typically used are first-order Euler implicit scheme for steady flows and the three-step backward method for unsteady flows. The other choices for steady flow problems are steady solvers with SIMPLE or artificial compressibility approaches.

4. NEW DIRECTIONS

4.1 New modeling techniques

There have been significant efforts in the marine industry to integrate CFD simulation capabilities over past several years. Accurate and fast simulation of turbulent free surface flows around surface ships and ocean structures has a central role in the optimal design of naval vessels and ocean structures. The flow problem to be simulated is rich in complexity and poses many modeling challenges because of the existence of breaking waves around the structures, and because of the interaction of the two-phase flow with the turbulent boundary layer. Some new numerical modeling techniques and trends for ship and ocean engineering flows are reviewed as follows.

LES (Large-eddy simulation)

LES is a numerical technique in which large scale energy containing eddies (those responsible for the primary transport) are resolved explicitly and only the small-scale sub-grid motions are modeled. The LES technique solves the unsteady three-dimensional Navier-Stokes equations with an appropriate filtering procedure. The filtered equations involve Reynolds stress type terms (contributions from the sub-grid scales) which are modeled by relating them to strain rates with an eddy viscosity as the proportionality coefficient. This is referred to as SGS closure model. The eddy viscosity is usually calculated as a function of the mesh size. Hence, the finer the numerical mesh size, the less important is the effect of the smaller scales that are filtered out. The three-dimensional time-dependent details of the largest scales of motion (those responsible for the primary transport) are computed. The size of the scales that need to be resolved determine the numerical mesh size to be used. LES makes extensive use of computer power rather than solving a large number of modeled equations as is the case for RANS models (Verma and Mahesh 2012). In this regard it also requires the use of accurate numerical schemes. LES is perceived as an effective tool for tackling and capturing the coherent turbulence structures near the free surface. The enhanced spreading of the wake near the free surface can be predicted well by LES. Much of the uncertainties in turbulence modeling can be eliminated if LES is extended to two-phase flow with appropriate modifications (Bhushan et al. 2013).

DES (Detached-Eddy Simulation)

Validation of closure models for the RANS equations has been an ongoing effort for several decades. These validation efforts are the key to obtaining a good description of the validity, accuracy, and utility of the various models over a range of applications. Flows involving massive separation and/or turbulent flow structure that scales with ship and ocean structure size comprise an especially difficult class of problems for RANS models. As available computing capacity increases, CFD researchers and practitioners are moving towards the use of LES as a higher fidelity alternative to RANS (Jang and Mahesh 2013). However, LES suffers from stringent near-wall spatial resolution requirements, and thus a practical alternative that seeks to leverage the best qualities of RANS and LES are the so-called hybrid RANS/LES methods, like DES, first proposed in 1997 (Spalart 2009). Generally speaking, a hybrid RANS/LES model applies a RANS closure model in the attached boundary layer region and a LES subgrid-scale model in regions of massively separated flow. The equations of motion are usually, but not necessarily, integrated in a time-accurate way for both the RANS and LES regions. The RANS and LES regions may be detected by a zonal scheme or a blending parameter (Sebastien 2011, Huang et al. 2012).

The validation of hybrid RANS/LES models is a difficult subject (Rui et al. 2013). RANS models are amenable to the usual verification/validation sequence; solution verification (grid refinement and iterative convergence criteria) is performed to assess numerical error in the solution. Then the model error may be assessed without complication. Conventional LES techniques are inherently difficult to verify and validate. Usually, the filter width is related to the grid spacing so that, as the grid is refined, the model and therefore, the solution, are also refined. This occurs simultaneously with numerical error reduction. The grid-refinement limit becomes direct numerical simulation which is, of course, impracticable for most flows of interest. Fixing the filter width and then applying grid refinement is a possible solution, but this strategy can be expensive and difficult to apply to complex geometries.

Lagrangian Particle Methods

There are two major Lagrangian particle methods: SPH (Smoothed Particle Hydrodynamics) and MPS (Moving Particle Semi-implicit) (Amini et al. 2011, Paredes and Imas 2011). Both methods use particles and calculate fluid behavior based on Navier-Stokes equations; however, the basic idea is different. SPH considers that the physical parameters in a particle, like mass density, velocity, etc., do not belong to the particle itself but are distributed smoothly around the particle, and a kernel function is used to calculate the physical properties of each particle (Kagatsume et al. 2011). On the other hand, MPS considers that the physical properties belong to the particle itself, and calculates the interaction between particles with weight functions (Khayyer et al. 2011). MPS can also be applied to incompressible flow by satisfying the condition of constant density. In addition, the solution process of the MPS method differs to that of the original SPH method as the solutions to the PDEs are obtained through a semi-implicit prediction-correction process rather than the fully explicit one in original SPH method. Improved versions of the MPS method have been proposed for enhancement of numerical stability, momentum conservation, mechanical energy conservation and pressure calculation (Gotoh 2012). The Lagrangian particle methods have proved their ability to capture physical features of violent fluid motions both around and on a vessel (Zhang et al. 2013).

Open Source Programming

Open source refers to a program or software in which the source code is available to the general public for use and/or modification from its original design free of charge. Open source code is typically created as a collaborative effort in which programmers improve upon the code and share the changes within the community. Not only the features of open source codes are comparable to the commercial CFD software, they also provide a perfect general and open platform for developing new numerical methods and tools (Yang et al. 2011). In recent years, open source codes such as OpenFOAM, FreeShip, Gerris, DUNS, have become popular; OpenFOAM is among the best. OpenFOAM is an object oriented C++ set of libraries for solving various partial differential equations using the finite volume method. It includes pre-processing, post-processing tools and specialized CFD solvers. Because of its object-oriented construction, users can implement codes for their own applications. Kawamura and Fujisawa (2013) used OpenFOAM CFD toolkit to form a new code which can simulate the flow around a self-propelling ship hull. NavalHydro-Pack is based on OpenFOAM and integrates core features of naval CFD into a set of optimized solver applications in ship hydrodynamics (Christ 2013). Several wave-makers including piston wave maker, flap wave maker and inlet wave boundary were implemented in OpenFOAM to numerically generate regular and irregular waves, directional waves, freak and rogues waves, focused waves, etc. (Cao et al. 2014). A dynamic overset grid capability has also been implemented in OpenFOAM aiming at ship flows for seakeeping, manoeuvring and ship-ship interaction (Wan et al. 2012, Shen et al. 2013, Shen et al. 2014).

GPU Parallel Computations

CFD solvers are among the most important applications that are run on supercomputers, consuming countless processing hours per year. The gap between the capabilities of the CPU and the complexity of the problems to be solved continues to widen. GPU has become an alternative platform for computing. There are a few codes in the CFD field that can benefit from GPUs or other similar parallel platforms. The result is a many-fold improvement in performance, allowing for more complex simulations in less time without sacrificing accuracy. In a potential future it is possible that CFD solvers can be designed for a GPU desktop, as well as those that run on the largest clusters. In some cases, the transition to GPU parallel computing requires little effort, but in most cases a full mathematical retooling is required. Though the potential is enormous, RANS-based codes have not been able yet of taking full advantage of GPUs for acceleration and modest improvements have been achieved. Compressible solvers and particle-based methods have seen better improvements. The limited success on GPU use in CFD is due to the low memory capacity in all GPUs.

4.2 Real time CFD

Ship manoeuvring simulators are an extremely useful tool for training of ships crews and for research of manoeuvring characteristics of ships. Usually, ship motions in the simulators are computed using equations of motion and hydrodynamic forces and moments required for these equations are obtained by experiments, database or prior computations including CFD.

With the increase of computing power, CFD analyses become faster than ever before, which develops the concept of real-time CFD computations linked to manoeuvring simulators. Pinkster and Bhawsika (2013) constructed the system which combines the real-time potential flow computation and the manoeuvring simulator for analysis of ship-ship and ship-port interactions. In the application, the potential code runs with the time step of 0.6 to 1.3 sec. while the time step of the simulator is 0.2 sec. Navier-Stokes solvers obviously requires more CPU time even with parallel computation techniques.

In this section, the feasibility of real-time CFD analyses linked to manoeuvring simulators is examined. CPU time of typical CFD manoeuvring simulations is estimated using the questionnaire of SIMMAN 2008 workshop (2008). The grid points are 3 - 5 million and time step size is 0.01 - 0.02 sec. for model scale computations. The average CPU time is approximately 1e-5 sec./iteration/points with the machines of 5 GFLOPS to 20 TFLOPS which yields one CFD iteration (0.02 sec. in real time) with 5 million grid points requires 50 sec. Therefore, in order to give the hydrodynamic forces and moments in real time, CFD must run 2500 times faster than present. Time step size may vary in full scale simulations and in case that ship-ship or other interactions are taken into account, CPU time increases significantly.

It is concluded that the use of real-time CFD methods linked to manoeuvring simulators are still beyond the capability of the present computing environment.

5. WAKE SCALING

5.1 Introduction

Wake scaling is a procedure or method to scale up a model scale wake to a full scale one by taking into account the influence of Reynolds number scale effects at the two different ship sizes. Wake scaling is usually needed in the following two situations:

(1) To extrapolate the measured model scale nominal wake to full scale, the goal of which is to provide a realistic wake field for propeller design;

(2) To produce a wake as close as possible to the full scale one in a cavitation test for small to medium size cavitation tunnels where it is impossible to accommodate a complete ship model, the aim of which is to improve the accuracy and reliability of experimental prediction of cavitation extent and pressure fluctuation in a cavitation test.

Traditionally wake scaling methods may be divided into two categories. The first is a simple wake contraction method where the “width” of wake is reduced in different manners. The second involves more complicated scaling steps based on boundary layer and potential flow theories, as reported for example by Tanaka (1979) and Sasajima (1966). The 26th Specialist Committee on Scaling of Wake Field has made extensive survey in this field.

5.2 Validation and use of CFD for wake scaling

CFD for wake scaling means the use of CFD (mainly RANS) methods to aid the development of a scaling method, as contrary to the concept of direct prediction of a full scale wake field using CFD methods. In a strict sense, the validation of a CFD method for wake scaling would inevitably involve two studies: a validation against the measured wake at model scale and a validation against the wake data at full scale.

There are many validations of RANS methods for wake prediction at model scale. However, validation at full scale is very rare, partly due to the fact that too few wake data from real ships are available and partly because full scale computations still present some challenges for RANS solvers.

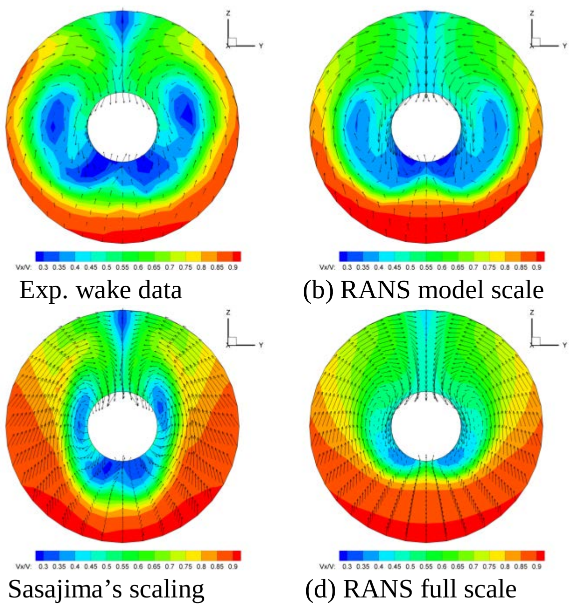
The results of Gothenburg 2010 Workshop on CFD in Ship Hydrodynamics (Larsson et al. 2014) are considered as representative of the state of the art in wake prediction for model scale ships. A total of 45 computed wake data at the propeller plane for three ship models were assessed and validated against the respective measured wake data, following established validation procedures. The number of submissions is by far the largest in the series of workshops, providing an invaluable database and statistics for the level of accuracy achievable for wake prediction by CFD methods today.

The assessment (Larsson et al. 2014) of this workshop results shows that: (a) The overall agreement between computations and experiments in terms of wake patterns at the propeller plane was fairly good; (b) The characteristic features of the bilge vortices were captured by the majority of CFD solvers; (c) The turbulence model has a profound influence on the accuracy of local flow structures. The flow details predicted by the advanced turbulence models like Reynolds Stress Model (RSM) and Explicit Algebraic Stress Model (EASM) show clearly better agreement with measured data than those simpler isotropic eddy-viscosity models, for the wake field of full form ships that has strong anisotropy Reynolds stresses and hook shape iso-contours of the axial velocity; (d) A 3~4 million grid on a half hull with a 2nd order discretization scheme seems sufficient to make a good prediction at model scale (without any appendage and free surface effect). The workshop organizers and participants seem to agree on the maturity of RANS methods for prediction of model scale wake field.

Some of the trends observed at the workshop were confirmed again by the later work of Wang et al. (2010) and Bull (2011).

A good example of use of CFD in wake scaling for propeller design was presented by Gaggero et al. (2013), in which the full scale wake of a research vessel was predicted by means of Tanaka-Sasajima's semi-empirical wake scaling method and a RANS solver. The resulting wake fields were then compared in terms of velocity values and the propeller unsteady cavitation behavior predicted by a panel method. The results revealed that the full scale wake field scaled by Sasajima's scaling method is significantly different from the direct RANS prediction (compare plot (c) with plot (d) in Fig. 5.1.1). The bilge vortex obtained by the scaling method is stronger and appears in a different location as compared with the RANS method. Study as such allows for exploiting of the differences that may be expected by the designer using the two different approaches. The importance of correct prediction of the wake velocity field in propeller design is emphasized.

Wake scaling is also utilized to improve the experimental prediction of propeller cavitation and induced pressure fluctuations in cavitation tunnel tests. A recent work is presented by Schuiling et al. (2011) and Wijngaarden et al. (2010). They utilized a RANS code to inversely design a model hull that generates a wake field closely resembling the full scale ship wake than does the geometrically similar ('geosim') hull model. As a demonstrator, a non-geosim scale model of a container vessel was manufactured and tested in a tank. The results were compared with available full scale data for correlation. It was concluded that the wake scale effects may largely explain the model to full scale correlation error on the blade rate hull pressure amplitude, and the use of a non-geosim afterbody design may correct for this sort of error to some extent, showing an improvement in predicting the 1st harmonics pressure pulses in the cavitation tunnel test.



(a) Exp. wake data (b) RANS model scale

(c) Sasajima's scaling (d) RANS full scale

Figure 5.1.1. Model/full scale wake predicted by a RANS method compared to Sasajima's scaling method and wake measurement data (Gaggero et al. 2013)

5.3 Determination of full-scale nominal wake by RANS methods

The maturity of numerical schemes and advancement of computing power has made it possible to directly calculate the wake field at full scale by RANS methods. EU project EFFORT is a project dedicated to improvement and validation of RANS methods for prediction of full scale viscous flow and wake field. Model and full scale experiment data for seven ships were made available for the consortium to study the accuracy level of wake prediction at full scale and to decide on the best turbulence models. Main achievements were reported by e.g. Verkuyl et al. (2003) and Starke et al. (2006).

Yang et al. (2010) evaluated the performance difference of a RSM and a Realizable k-ε model (RKE) for nominal wake at model and ship scale for three hull forms. It was found that the RSM provided much better agreement with wake measurement data at model scale in terms of boundary layer thickness, hook shape and iso-wake contours, and that the RSM model seems to be better at full scale as well.

Choi et al. (2011) compared cavitation patterns of model tests with those from a full-scale sea trial for a VLCC propeller that suffered from erosion near the tip region. Cavitation tests were performed at design and ballast draft, using model and full scale nominal wake, respectively. A model ship and a wire mesh method were used for the simulation of wake patterns of model nominal wake. For the prediction of full-scale wake field, a RANS solver was employed and a wire mesh method was used for the simulation of the full scale wake. The results show that cavitation patterns and the location of cloud cavity in relation to the eroded area are closer to those observed in the sea trial at ballast draft, when using the RANS-predicted full-scale wake field in the cavitation test.

A new trend in full scale wake prediction is for ships equipped with Energy Saving Devices (ESDs). The application is of particular interest for ITTC society as the scale effects on ESDs are less explored in routine towing tank tests and cavitation tunnel tests. Hopefully CFD studies may provide some invaluable information on the scaling trends of ESDs. Heinke et al. (2011) investigated the Reynolds number scale effect on the flow around a wake equalizing duct of Schneekluth design (WED) and vortex generators (VG) and on the inflow to the propeller. The investigation showed that knowledge of full-scale wake is necessary for the accurate design of propeller and the prognosis of cavitation behaviour and propeller induced pressure fluctuations for ships with WED and VG. Kim et al. (2012a) compared the model and full scale nominal wake difference for an Aframax product carrier equipped with Pre-Swirl Stators (PSS) using an EASM turbulence model.

Likely challenges for full scale wake calculations are the extremely high aspect ratio of grid cells, large grid size and lack of validation of turbulence models at full scale. The latter is partly due to a general lack of full scale data. The current practice is to perform some form of validation study at model scale, then assume that conditions will be similar and apply the same or similar grid strategy, discretization scheme and turbulence model for full scale calculation without further validation.

5.4 Full scale wake data

There have been some efforts in measuring full scale wake, for example, in project EFFORT (Verkuyl et al. 2003), Euclid 10.12 (Di Mascio et al. 2001) and DALIDA

(Perelman et al. 2012). However, wake data from these projects are not publically available. A report containing full scale measurement data for the Athena Research Vessel (Day et al. 1980) is available.

5.5 Determination of effective wakes by RANS/potential-flow coupling

The effective wake is the nominal wake as deformed by the interaction with the running propeller with exclusion of the propeller self-induced velocities. The effective wake is estimated by coupling a potential-flow method for the simulation of the propeller with a RANS solver for the simulation of the bulk flow around the ship hull.

The literature on the estimation of effective wakes has grown in the last few years. Villa et al. (2011), use both an actuator disk model and a three dimensional body force approach. Kinnas et al. (2012) present a conservative interpolation scheme for the body forces and uses curved zones in front of the propeller leading edge or blade control points as locations for the velocity extraction in the RANS/potential-flow coupling.

Starke & Bosschers (2012) discuss RANS/BEM coupling errors in the prediction of the effective wake especially at the propeller root. They suggest calculating the induced velocities only from the dipoles, not from the sources.

Rijpkema et al. (2013) tackles the problem of the location of the extraction velocities and suggests calculating the velocities on two planes upstream the propeller and extrapolating them to the propeller plane. Alternatively, curved axi-symmetric surfaces in front of the leading edge can be used. Large errors in the prediction of effective wakes are expected in the presence of separated flow.

In Sánchez-Caja et al. (2014), special attention has been paid to the errors in coupling RANS with potential flow methods. A correction factor approach is developed suggesting that the error derived from coupling a potential flow method for the representation of the propeller with a RANS solver can be split into two parts: one that is common to any propeller potential method due to the fact that such potential method is or should be a good approximation to the propeller viscous solution, and the other one which is dependent on the particular potential method used. The first part, represented by the correction factor approach is a first order correction to the potential flow solution. It can be calculated in a simple setup (propeller in uniform flow) and used in a complex one (propeller in ship wake). Also the paper proposes a concrete dependence of such correction on the local induced flow. Further development of this approach may be a step forward for the calculation of effective wakes in extreme situations (for example propeller under oblique flow or high loadings).

5.6 Conclusions

Model scale nominal wake can be predicted fairly accurately with more advanced anisotropic turbulence models (e.g. EASM and RSM) and the SST k-ω model with curvature correction. Full scale wake prediction is achievable, provided that sufficiently fine grids are used to resolve the near-wall boundary layer and appropriate turbulence models are employed. Experience seems to suggest that prediction differences between advanced turbulence models and eddy viscosity models becomes smaller at full scale because of the reduction of anisotropic Reynolds stresses (i.e. disappearance of the “hook”). Following the

27th ITTC Practical Guideline for RANS Calculation of Nominal Wakes (7.5-03-03-03), RANS methods can be a useful tool to deal with scale effects in wake field.

6. VERIFICATION AND VALIDATION

6.1 Introduction

Assessing the accuracy and uncertainty of numerical solutions has always been a challenging task but this has become an impellent necessity since the dramatic growth of computational resources and the consequent increased use of numerical codes as design tools. Different procedures and standards have been developed for this purpose; some of these are applicable to any numerical code while others are for specific applications. Between the most popular verification and validation (V&V) guidelines, there are the ASME guide for V&V in Computational Solid Mechanics (ASME V&V10) and the ASME standard for V&V in CFD and Heat Transfer (V&V20). Specifically for CFD applications, also important are the AIAA guides for V&V in CFD (AIAA G-077-1998) and the various ITTC guidelines on V&V (7.5-03-01-01, 7.5-03-01-02, 7.5-03-01-03, 7.5-03-01-04). A review of research papers in V&V for ship hydrodynamics can be found for instance in Roache (1997), Eça and Hoekstra (2012) and Larsson et al. (2014).

6.2 Verification of calculations

Verification of calculations is the procedure to estimate the numerical error and uncertainty of computed results. As proposed by Roache (1997), verification can be described as “solving the equations right”, as opposite to validation which is “solving the right equations”. Importantly, verification should be performed for each quantitative result extracted from the numerical solution, i.e. the so-called quantities of interest, such as for instance the global resistance of the ship, the local pressure and flow velocity.

Different verification procedures are available for the estimate of the numerical uncertainty. Most of the available procedures aim at computing the numerical uncertainty within a 95% confidence level. The probability functions are assumed to be Gaussian and centred in the solutions, therefore the uncertainty is defined as twice the standard deviation. In other words, there is 95% probability that the solution is within the range between the computed value subtracted of the numerical uncertainty and the computed value added of the numerical uncertainty.

The numerical uncertainty is an amplifying function of the magnitude of the numerical error, which is the difference between the computed result and the exact solution of the chosen equations. The numerical error is typically broken down into the errors due to different sources: a finite time and space discretisation (as opposite to a continuum), a finite number of iterations (leading to the iterative error), the round off of the numbers and other sources. Assuming that these errors are independent to each other, the total error is computed as the mean square root of the errors. This approach is not universally accepted and, for instance, Eça and Hoekstra (2006) recommends that the iterative error is linearly added to the mean square root of the other errors because the iterative error is not independent from the discretisation error.

The estimate of the error is typically based on the trend of the results for different values of the parameter that mostly drives the source of error. The results are extrapolated for the value of the parameter that should lead to a null error. For instance, assuming that the discretisation error depends from the mean node distance, then different mean node distances are tested and the results are extrapolated for a null mean node distance. For each mean node distance, the discretisation error can be computed as the difference between the result and the extrapolated value for a null mean node distance. Similarly, the round off error can be estimated as the difference between the result and the extrapolated value for an infinitely accurate machine using an infinite number of digits, etc.

6.3 Different verification methods

The various existing verification methods differ for how the extrapolation is performed and for the procedure used to compute the uncertainty from the error. All the different extrapolations methods are based on power series expansions and are equivalent when the solution monotonically converges toward the exact solution with the expected order of convergence and without scatter. Larsson et al. (2014) grouped the most common verification methods in ship hydrodynamics into three groups: the Grid Convergence Index methods (GCI), the Factor of Safety methods (FS) and the Least Squared Root methods (LSR). All these methods have different limitations and none of them has been recognised as superior to the others.

The GCI, mostly developed by Roache (1998, 2009), requires the solution to converge monotonically and the Richardson extrapolation (Richardson, 1911) is used to compute the solution with zero error. Unfortunately the set of results does not always converge and, in such cases, this method cannot be used. When convergence occurs, the error is computed as the difference between the computed result and the extrapolated value. The uncertainty is the product of the error and a factor of safety. Roache recommends a factor of safety of 3 or 1.25 depending if two or more values of the parameter are used to compute the Richardson Extrapolation.

The FS method was mostly developed by Xing and Stern (2010) and it was used to develop the ITTC guidelines. It shares with the GCI the limitation of requiring monotonic convergence in order to perform the Richardson Extrapolation. This method focuses on improving the estimate of the factor of safety which is defined as a function of the ratio between the effective and the theoretical order of convergence. Unfortunately the theoretical order of convergence, which for the discretisation error is the order of accuracy of the numerical algorithms implemented in the code, is not always known. Most of the CFD codes in ships hydrodynamics are second order accurate in space when Cartesian grids are used; and the observed order of convergence is typically between one and two. The use of the FS method leads to a minimum factor of safety of 1.6, which is achieved when the observed and theoretical orders of convergence coincide.

The LSR method was proposed by Eça et al., (2010a, 2010b). It uses a LSR fit for the extrapolation of the zero error solution. This approach is very interesting because it allows estimating the uncertainty also when a converging trend is not achieved. The uncertainty is a function of the standard deviation of the LSR fit, the observed order of convergence and the distance to the extrapolated solution. Viola et al. (2013) also suggested that the uncertainty should decrease when the number of results used as input to the LRS fit increases, and when a wider range of the parameter is explored. Recently Eça and Hoekstra (2014) expanded further this method considering different truncated power series expansions and adopting weights to increase the influence of those solutions nearer the extrapolated values, to increase the influence of those solutions achieved with fine grids.

6.4 Validation

The validation aims at assessing the choice of the numerical model as a representation of the reality. Validation is typically made against a physical model, where the physical solution is known within a given experimental uncertainty, and the input parameters (i.e. the Reynolds and Froude numbers, the turbulence intensity, etc.) are known within an input uncertainty. The experimental input and numerical uncertainties are combined with the mean square root in order to compute the validation uncertainty. The numerical result is said “validated at the level of the validation uncertainty” if the difference between the numerical and the physical result is smaller than the validation uncertainty. It should be noted that the emphasis should not be on the success of the validation, but on the level of validation uncertainty. In fact, the higher the uncertainty (and thus the poorer the quality of the result), the most likely the result would be validated, but it would be validated at the level of a higher validation uncertainty. The user should verify if the achieved level of validation uncertainty is sufficiently small compared to the differences that are objectives of the CFD investigation, such as for instance the differences between different design candidates or the differences between different operating conditions.

6.5 Conclusions

The present review shows the significant effort of researchers to find a reliable, accurate and robust method to perform V&V studies for CFD computations. Unfortunately this seems far from having been achieved and often these objectives are incompatible to each other. One of the most important challenges of V&V is to become a common practice for the industry. This can be achieved if V&V will become easy to perform, inexpensive and its results easy to be interpreted. In fact V&V is often regarded as unaffordable from industrial users because it requires too many resources and provides too little information. Often, a large number of simulations are performed and a wide range of quantitative results is gathered from each simulation, making impractical to perform V&V for each of those results. On the contrary, if few specific quantities of interest are identified, V&V is an affordable and an essential tool for the interpretation of the CFD results and to improve the numerical model. In fact, being CFD results approximations and not exact solutions, those are meaningless without knowledge of the associated uncertainty. While hopefully future research will allow developing a more intuitive and less computationally demanding methods, there is an urgent unmet need to make the industry more aware of the potential benefit of V&V and to inform CFD users on how the V&V results should be interpreted.

7. TRENDS IN NAVAL ARCHITECTURE APPLICATIONS

This chapter summarizes ongoing research efforts toward the development of efficient numerical tools in the area of computational hydrodynamic analysis for ships, submarines and other water craft, reporting trends in research and experience in industrial applications as emerged from the literature of recent years. The section outlines the trends that have been observed in each of the traditional naval architecture areas: Resistance, Propulsion, Propellers, Seakeeping, Manoeuvring, and Ocean Engineering.

7.1 Resistance

The past three years since the 26th ITTC have seen a continuing progress in the area of resistance. The trends in use of CFD for resistance applications discussed in the last ITTC report are still evolving. Among others, the size of the computational grids used by typical CFD practitioners has kept growing, reaching up to several tens of millions of elements. High-performance computing (HPC) on parallel machines with thousands of cores helps reduce solution turnaround times. We continue to see proliferation of unstructured grids for real-world's ship applications involving complex geometry. Nonetheless, complex flow physics carried by turbulent flows and free-surface flows (waves) still pose significant challenges.

7.1.1 Review of recent literature

Resistance of a ship is the first and foremost quantity of interest when speaking of ship hydrodynamic performance. Resistance prediction is the most mature of all CFD applications in ship hydrodynamics.

The status of the matter is timely and concisely exposed in the book recently published under the title of "Numerical Ship Hydrodynamics" (Larsson et al. 2014) that gives an assessment of the Gothenburg 2010 Workshop results. We here recapitulate the summary and conclusions in the area of resistance for KVLCC2, DTMB-5415, and KCS, the three cases selected for the workshop.

The CFD codes used by the participants varied widely, including in-house and academic codes (ISIS, WAVIS, NavyFOAM, SURF, FINFLO, to name a few), and commercial codes (FLUENT, STAR-CCM+, CFX, SHIPFLOW). There were multiple submissions from different groups using the same commercial codes, most notably with FLUENT and STAR-CCM+. This reflects the popularity of commercial CFD packages.

The majority of the participants used variants of 'projection methods' or what may be called 'pressure-based methods' such as SIMPLE and PISO to satisfy the continuity equation and to advance the solution in time. The rest employed coupled solvers based on artificial compressibility. For spatial discretization, finite-volume method (FVM) with formally second-order accuracy was predominantly adopted. However, as in the past, there were no submissions using finite-element (FE) codes. More than half of the participants employed unstructured grids, although the relatively simple geometries of KVLCC2, KCS, and DTMB 5415 could be gridded up as easily using (multi-block) structured grids. This seems to indicate that an increasing number of CFD practitioners in ship hydrodynamics prefer unstructured grids mainly due to ease of meshing and time-saving they offer.

The majority of the contributions employed isotropic eddy-viscosity models (EVM), mainly the family of k-□ models. Some contributors opted for EASM. However, their predictions were only equally good or marginally better than EVMs in the predictions of resistance and characteristic features of the mean axial velocity in the hull boundary layer and at the propeller plane. There were no contributions based on differential Reynolds-stress models (DRSM), although the efficacy of DRSM to accurately capture cross-flow separation on and ensuing vortices giving "hook-like" mean velocity contour at propeller plane was demonstrated more than a decade ago at the 2000 Gothenburg Workshop (Kim and Rhee, 2002). In terms of wall boundary condition, the majority resolved all the way down to the viscous sublayer, explicitly applying a no-slip condition at wall. Several participants, however, relied on wall-functions approach that alleviates near-wall resolution requirements, especially for full-scale ships.

The 2010 Gothenburg workshop saw submissions using LES (FOI using OpenFOAM). There were also contributions using RANS/LES hybrid approach (IIHR using CFDSHIP-IOWA). The rationale behind and the potential benefits of LES and hybrid RANS-LES have been recognized by the ship hydrodynamics community. At a significantly higher computational cost than RANS, LES when properly executed can directly resolve large-scale, turbulent coherent structures. The main roadblock inhibiting adoption of LES is the prohibitively high computational cost that is due to extremely fine resolution of grid required to properly resolve the length- and time-scales of the energy-containing eddies down to the "inertial subrange". Thus, 'legitimate' LES satisfying this resolution requirement is still beyond the reach for resistance prediction of ships with largely well-attached, high-Reynolds number turbulent boundary layers. Under-resolved LES can give poor prediction of viscous resistance as demonstrated by Alin et al. (2010). The prospect is brighter with the RANS/LES hybrid approach, inasmuch as the hybrid model supposedly should switch to RANS mode somewhere in the boundary layer, which will help retain the ability of RANS to predict resistance. However, due to the well-known sensitivity to near-wall grid for some hybrid RANS-LES methods, precautions and careful validations are necessary before the hybrid approaches can be relied upon.

As for the method to resolve the air-water interface for surface ship applications, the contributors were found to split almost equally between VOF and LS approaches. The strengths and weakness of these two approaches are well known and discussed in the literature. In terms of performance (accuracy and stability), they seems to be largely on par, as far as resistance applications are concerned.

Among the major conclusions of the Gothenburg 2010 Workshop are:

- The mean comparison error, defined as the average over all three cases and all the submissions of the difference between the measured value (D) and the predicted value, was a mere -0.1%. The mean standard deviation was 2.1%.

- Grid-convergence of the resistance prediction was demonstrated roughly by a half of the all submissions, mostly by the participants who adopted structured grids. There was only one grid convergence study of all submissions employing unstructured grids. One concern was that the order-of-accuracy derived from the computations on systematically refined grids was found to be inconsistent with the formal order-of-accuracy of the discretization schemes employed.

- The errors and standard deviations of sinkage and trim were larger than those for resistance prediction, particularly in the low speed range. The larger errors are ascribable to both experimental and numerical difficulties of measuring sinkage and trim accurately and stably at low speeds. For Froude numbers above 0.2, the mean comparison error was found around 4%, and the standard deviation around 8%.

- Wave patterns around all three hulls were predicted with fair accuracy, with the wave cuts near the hull better predicted than those farther away from the body. The accuracy of the predictions farther away from the hull varied widely mainly due to the differing grid resolutions used in the computations.

- Turbulence modeling has little effect on the prediction accuracy as far as the resistance is concerned. The predictions with advanced turbulence models such as explicit algebraic Reynolds-stress models did not show appreciable improvements over those obtained using two-equation, eddy-viscosity models.

- Wall function approaches skipping the viscosity-affected near-wall region did not seem to compromise the quality of the solution, doing a commendable job in predicting resistance and resolving boundary layer and secondary flows.

Better yet, the statistical variance (scatter) of all the predictions submitted by the participants was substantially smaller than had been found in the previous workshops in 2000 and 2005. The smaller scatter might be attributed to participants' collective learning made over the years on those widely known test cases. Still, it can be hailed as a progress. Thus, one can say that, at least for types of ships and their operating conditions akin to the ones computed in the workshops – mono-hull without appendages on a straight ahead operation, the fidelity of CFD for resistance prediction has now reached a level that comfortably exceeds, at least, what is considered sufficient as a design tool.

For unconventional ships such as multi-hulls, planning boats, and new-concept hulls, it is a little harder to assess the state of the matters due to scarcity of relevant publications.

Haase et al. (2012) used a RANSE-based method to predict the resistance for medium-speed catamarans with various hull forms, speeds and scales using both an open-source RANS solver and a commercial solver. For the Froude numbers of interest ($0.3 < Fr < 0.5$), they found that mean prediction error of less than 10% could be achieved compared to the model- and full-scale test results.

It is interesting to note the use of smoothed particle hydrodynamics (SPH) to compute the free-surface flow around a planing hull (Dashtimanesh and Ghadimi, 2013). The SPH approach using sub-particle scale turbulence model was validated for a transom flow. The results were in good agreement with experimental observations in the rooster tail.

Lastly, it is worth noting a new approach based on a combination of RANS method and potential-flow theory that shows a potential to significantly cut down the solution turnaround time for calm resistance predictions (Rosemurgy et al., 2011). This method utilizes a velocity decomposition method, and employs a combination of potential-flow approach and RANS method. Savings the computational time are achieved first due to the fact that a RANS domain much smaller than the ones used for traditional RANS computations can be used. Furthermore, single-phase RANS approach can be used, since use of linearized free-surface boundary conditions allow the field discretization to extend only below the calm free-surface plane. The method was demonstrated for the Wigley hull and DTMB 5415 model. The predictions agree commendably with experiments. The time to complete computations of the total resistance on a ship was found to be a small fraction of that required by full RANS methods.

7.1.2 Benchmark cases

For calm water resistance, KCS, KVLCC2 and DTMB 5415 were used for the Tokyo 2005 and Gothenburg 2010 CFD Workshops. KCS is a container ship, KVLCC2 represents low-speed, tanker ship with high blockage-coefficient, whereas DTMB 5415 is a medium-high speed surface combatant. Complete geometries and datasets for these geometries are available online. A large variety of other geometries are available with resistance data, including Wigley hull, Athena (research vessel), Series60, S175, ONR Tumblehome, HSVA Tanker and Mystery Tanker (Dyne Tanker), Ryuko-maru (tanker), Seiun-maru (training ship), Hamburg test case (container ship), and Duisburg test case (container ship), to name a few.

7.1.3 V&V procedures and needs

For resistance, V&V procedures are well established (Larsson et al. 2014, Carrica et al. 2011). Some CFD practitioners use their own V&V procedures tailored to their practices. Ideally, it would be nice if an ITTC-endorsed V&V procedure can be developed and used among the CFD practitioners. No matter what V&V procedures are followed, we strongly recommend that they be continued. For ship designers, uncertainly quantification (UQ) would be of much interest.

7.2 Propulsion

Propeller-hull interaction on ships in a steady, straight-ahead course at or near a self-propulsion point continues to be the core issue in the area of ship propulsion. A ship propeller operating behind a hull alters the flow-field near its stern, locally accelerating the flow, which contributes to the 'effective wake". In addition, it lowers stern pressure, which results in augmentation of resistance also known as thrust deduction (See Fig. 7.2.1 illustrating the effects of a rotating propeller on the hull stern pressure). A propeller operating in a ship wake generates unsteady forces (axial and transverse) and moments due to the effects of non-uniform wake, a product of the upstream boundary layer over hull and appendages, and ensuing vortices emanating therefrom. The unsteady loading on the propeller exerts fluctuating pressure on the stern and excite hull vibration. It can also act as a source of noise. Interaction between propeller and rudder is often of concern as well, inasmuch as rudders are in the propeller slipstream with axially accelerated flow and added swirl and vortices, and as the rudder blocks and cuts though the propeller slipstream.

It is useful to note that propeller-hull interaction is arguably an even more important issue in maneuvering, inasmuch as it affects the accuracy of CFD predictions of maneuvering characteristics.

Thus, propulsion involving both hull and propellers is significantly more complex and harder to tackle using CFD than resistance, being compounded mainly by the presence of rotating propellers behind ship hull.

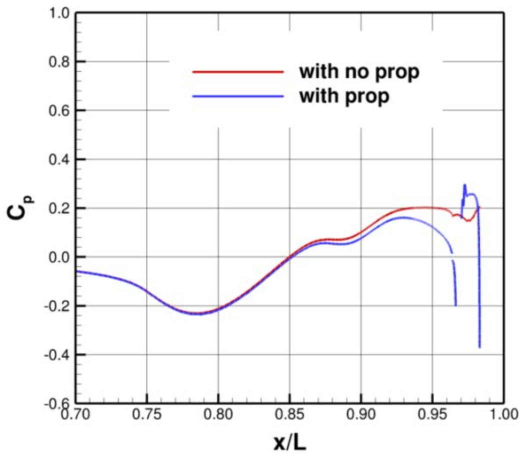
7.2.1 Review of recent literature

The fidelity with which the details of propeller-hull interaction can be predicted by CFD is determined largely by how accurately the ship wake – the inflow to propeller – and the highly complex three-dimensional flow around rotating propellers can be resolved. Thus, both minimization of discretization error and turbulence modeling continue to play important roles in determining prediction accuracy, even more than it does for resistance. Like in other ship applications, RANS computations are the main workhorse for propulsion applications. RANS computations to study propeller-hull interaction in earnest are bound to be still quite CPU-intensive, since accurate resolution of turbulent boundary layer over entire hull and wake requires a very fine grid and consequently a large amount of computational resources especially if time-accurate solutions are sought.

Although RANS is predominantly employed, LES and RANS-LES hybrid approaches have begun to appear recently in the literature for propulsion applications (Liefvendahl and Troëng, 2011; Castro et al., 2011; Chase et al., 2013). The rationale behind these much costlier approaches is that large-scale, turbulent coherent structures ingested into the propeller can be directly captured by computations using LES. However, at the time of writing this report, LES for simulation of propeller-hull interaction is, at best, at an exploratory and experimental stage. Legitimate LES with proper grids resolving down to the 'inertial sub-range' and near-wall region is still far from being feasible due to the prohibitively high cost required to adequately resolve the length and time scales required, and to extract stable statistics of engineering quantities such as resistance and time-averaged flow-fields. Hybrid RANS-LES approaches are not foolproof either due to their well-known high sensitivity to the grid resolution in near-wall regions.

The foremost issue with CFD computations for propulsion applications is centered around how to represent the effects of rotating propeller(s). Various approaches were reviewed in the last ITTC report. Their complexity, fidelity, and economy vary widely among the methods.

The 'body-force' approach seems to be the most popular among CFD practitioners due to its economy and reasonable accuracy. The methods under the banner of body-force approach differ from one another in the way the spatial distribution of body-force (momentum source) is determined. The simplest body-force method is one in which the classical 'actuator disk' model and its variants are used to compute the radial distribution of body-force in axial and tangential direction. The most sophisticated body-force approach today employs boundary-element method (BEM) to compute the momentum source distribution in the volume swept by propeller (Krasilnikov et al., 2013; Queutey et al., 2013; Chase et al., 2013; Rijpkema et al. 2013). The body-force approach, with the aid of today's computing power, should be affordable and perhaps sufficient for many CFD practitioners in design offices, model basins, and shipyards. It should be kept in mind that fidelity of any body-force approach can be only as good as the models constituting the approach, and may significantly degrade for some off-design conditions that are outside of the region of validity of the models.



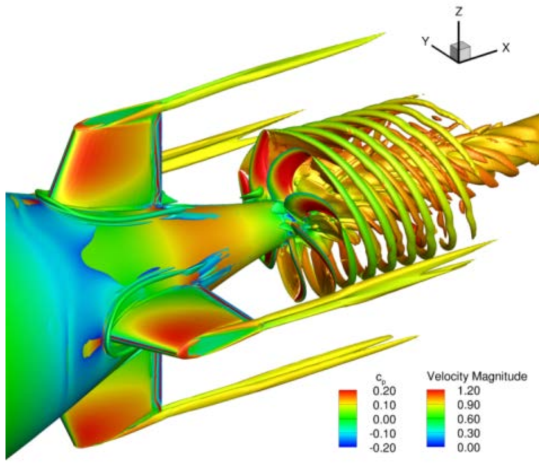


Figure 7.2.1. Effects of a rotating propeller – Top: Surface pressure with and without a rotating propeller; Bottom: Schematic view of the stern flow and the propeller slip stream J = 1.1, β = 0°.

It has become feasible in recent years to include a rotating propeller explicitly in CFD computations using sliding-grids or overset-grids techniques widely available in many off-the-shelf commercial CFD software packages and in-house codes (Muscari and Di Mascio, 2011; Carrica et al. 2012b; Gao at al., 2012; Chase et al., 2013). This 'direct approach' is far more – at least by an order of magnitude - compute-intensive and time-consuming than the body-force approach. And yet, the 'return of investment' is remarkable, inasmuch as it provides time-accurate predictions of the flow details associated with propeller-hull interaction including the unsteady loadings on individual propeller blades and propeller as a whole, and time-resolved flow-fields (velocity, pressure) on and near the stern, and in the vicinity of propellers. The wealth of the flow data resolved in space and time with this level of details is also extremely expensive to come by experimentally. Besides directly simulating interaction between propeller and hull using first principles, another important utility of the extensive dataset from the direct approach is that it allows us to develop or refine, and validating simpler 'models' for rotating propellers. The major hurdle at this point for CFD practitioners is the lengthy solution time. With large size of grids commonly used these days, the solution turnaround time for the direct approach is largely determined by the algorithmic constrains such as the allowable time-step size and computational efficiency of sliding-grids and overset-grids algorithms, especially their scalability on multi-core computers. It is hard to find information in the literature on the aspect of parallel efficiency of sliding-grids and overset-grids techniques.

The 2010 Gothenborg CFD Workshop (Larsson et al, 2013) had selected the KCS model (Case 2.3a) – a Korean container ship model - as one of the benchmark cases to evaluate capability of CFD today to predict propeller-hull interaction at a self-propulsion point. The contributions, a total of thirteen submissions for this case are split roughly into equal halves in adopting either the actual propeller (direct approach) and 'modeled' propeller (body-force approach). As far as the predictions of time-averaged K_t and K_q are concerned, the mean comparison errors were 0.6% and -2.6%, respectively. The standard deviations were found to be 7% and 6%, respectively, which is considerably larger than that for the resistance predictions. This is not surprising considering that the additional uncertainty is introduced by the different approaches to accounting for the effects of the rotating propeller. The improvements in the predictions of the time-averaged K_t and K_q using the direct approach were found to be marginal, despite the much smaller deviations.

The mean velocity fields predicted and submitted by the participants for Case 2.3a were also looked at and compared against the measured ones at the propeller plane ($x/L = 0.9825$) and on a plane in the propeller slipstream ($x/L = 0.9911$,). At the propeller plane where the measurements indicate presence of a fairly strong swirl in the mean flow, notable discrepancies in the axial and the transverse velocity components were observed between the predictions and the measurements. The direct approach did not seem to improve the predictions. At $x/L = 0.9911$, the measurements show a characteristic, asymmetric distribution of the mean axial velocity with “moon’s crescent-like” region of high velocity apparently caused by the axial acceleration and swirl generated by the propeller. Considering the complexity of the flow, the results obtained by most of the participants are considered to be in fair agreement with the experimental data. The majority of the predictions based on the direct approach and the body-force approach with the exception of too simple a body-force model captured the asymmetry and the region of high velocity. Again, the predictions based on the direct approach did not give substantially better results than those from the simpler approaches.

Despite some evidence indicating no improvement from the direct approach in the predictions of K_t, K_q, and the mean velocity distributions near the propeller, one should not write off the direct approach, because several factors, including the insufficiently accurate prediction of the upstream flow over the hull, are responsible for the discrepancy observed between the predictions and the measurements.

7.2.2 Benchmark cases

The KCS model, which was selected as the test case for a self-propelled ship in the 2010 Gothenborg CFD Workshop (Larsson et al, 2013), is still considered a good benchmark case, since a considerable gap exists between the predictions and the measurements. Since this case involves complex flow physics including the free-surface, the turbulent boundary layer over the hull and the propeller blades, the swirl and vortices generated by the propeller, and the propeller-hull interaction. A systematic grid convergence study is recommended, for it will help us to separate numerical discretization error from modeling error and to understand the main cause for the discrepancies.

In addition, the upcoming workshops dedicated to applications of CFD to ship hydrodynamics such as the 2015 CFD workshop in Tokyo may provide useful benchmark cases. Data collected for the latest CFD Workshop are also available, but, as concluded by the organizing committee, experimental uncertainty analysis, including facility bias, is still an issue.

7.2.3 V&V procedures and needs

For propulsion applications, V&V can be done on quantities such as K_t and K_q, thrust deduction, and effective wake. V&V is then conducted for these quantities as for the resistance in calm water (see for example Larsson et al. 2014, Carrica et al. 2011).

7.3 Propellers

7.3.1 Review of recent literature

As part of SMP-2011, a CFD-workshop was held on the computation of non-cavitating and cavitating flows. There were two test cases: the so-called Delft Twist-11 Foil and Potsdam Propeller Test Case. Five parties submitted results for the twisted foil and 14 groups the propeller case (Abdel-Maksoud, 2011).

Participants of the foil case were requested to simulate the flow around the foil in the tunnel under uniform inflow conditions. Conditions were specified for two cases: flow with and without cavitation. Hoekstra et al. (2011) summarise the computational results of the foil case: “Numerical problems in simulating cavitation of foils with RANS, DES or LES still remain. This is partly reflected by the observation that different people with different codes, but solving essentially the same mathematical problem (same type of equations, same turbulence model, same cavitation model), do not produce similar results.”

For the propeller three different cases were asked to compute: the open water performance, the velocity field at various measuring planes and cavitation at three different working point. Calculations from 14 groups were obtained for the evaluation of the open water characteristics, employing 10 solvers for viscous flow and 5 solvers for potential flow. The total number of submitted calculations is 19. Calculations from 11 groups were obtained for the evaluation of the cavitating propeller, employing 7 solvers for viscous flow and 5 solvers for potential flow and submitting 15 calculations.

The open water performance was asked to calculate at five different advance coefficients. There were quite large variations on the results. When comparing the calculated values to the measured ones the average difference were: K_t: −2.9% , K_q: 1.0% and η_0: −3.8% . The standard deviations were K_t: 3.3%, K_q: 3.6% and η_0: 2.8% . The maximum relative differences were K_t: 4.8% , K_q: 11.1% and η_0: 4.9% and the minimum K_t: −10.5% , K_q: −6.9% and η_0: −11.2%. An example of the relative difference is shown in Fig. 7.3.1.

In addition to the computations, there was a questionnaire of the computational non-viscous and viscous methods. All viscous flow methods were FV-methods. Unstructured grids were more popular than structured grids (10/3) and in coupling of the rotational part to the fixed part multiple reference frames were more popular than sliding mesh techniques (9/3). In the cavitation modelling all groups used mass transfer models, wheher the Kunz’s, Sauer’s and Singhal’s models were the most popular.

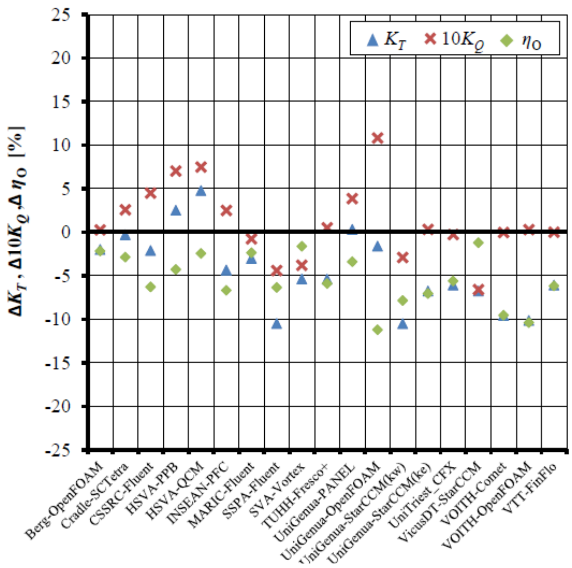


Figure 7.3.1. Relative difference between measurement and calculation, $J = 1.2$ (Abdel-Maksoud, 2011).

In their work Morgut and Nobile (2012) analyzed the influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers, working in uniform inflow. The study was carried out comparing hexa-structured meshes with hybrid-unstructured meshes using the SST turbulence model and the BSL-RSM turbulence model. The simulations were carried out with a commercial CFD solver. The numerical results were compared with the available experimental data of propeller P5168 and E779A in model scale. The comparison was carried out comparing the propulsive characters, evaluating the global field values, represented by the thrust and torque coefficients, and also considering some local field values measured in the propeller wake. Their computational results suggest that, for the numerical predictions of the propeller open water propulsion characteristics, the hexa-structured and hybrid-unstructured meshes can guarantee similar levels of accuracy. Nevertheless hybrid-unstructured meshes seem to exhibit a more diffusive character than hexa-structured meshes, and thus the former are less suited for detailed investigations of the flow field. Finally, the two different turbulence models behaved similarly on both types of meshes, with the BSL-RSM turbulence model providing only slightly better predictions than the more economical SST turbulence model.

Lindau et al. (2012) have modelled an axial flow waterjet (AxWJ-2) in water-tunnel test configuration using a powering iteration methodology. The flow was modelled over a range of conditions including cavitation breakdown. The single- and multiphase flow solutions appeared to accurately capture the integrated performance at all conditions. In addition, the overall cavitation patterns, on rotor blade suction surface and due to tip-gap flows, were well captured at a range of cavitation conditions. For ducted devices, such as waterjets, it is suspected that flow breakdown will coincide with cavitation choking in the rotor passage. This is suspected in the experimental data and has been demonstrated in the computational results of their work. It was shown that the present computational approach is useful and accurate when properly applied to the modeling of blade cavitation patterns and cavitation-driven thrust breakdown for axial flow waterjets.

Lu et al (2012) studied the unsteady cavitating flow around a 10 degree tilted marine propeller. They simulated the flow field with three computational methods, ranging from lifting surface methods and RANS to LES, to demonstrate the capability of different simulation tools for this complex flow. Their results indicated that although potential flow solver can predict fairly well the thrust and torque coefficient, and usually captures simple types of sheet cavitation, it cannot predict more complex sheets or root cavitation. RANS-method partly captured the dynamic evolution of the sheet close to the tip region and the occurrence of the root cavitation, however it predicted a leading edge sheet that is not present in the experiment. The missing of the vortical structure on blade limits also the use of RANS in analysis of some of the hydrodynamics that is crucial for understanding and controlling the cavitation and related noise and erosion. Their LES computation showed the tendency in filling in this gap by capturing the correct location and dynamic behaviour of the vortical structure mentioned above.

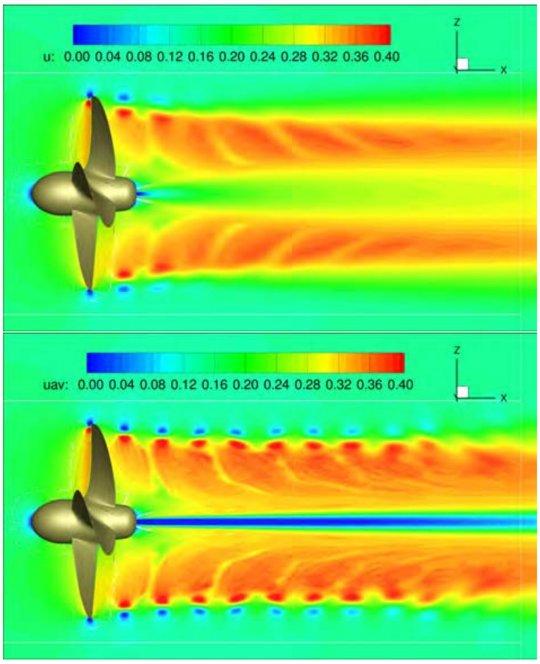


Figure 7.3.2. Comparison of two different turbulent flow approaches at J = 0.45. Section x-z of axial velocity. RANSE (top) and DES phase average (bottom).(Muscari and Di Mascio 2013).

Muscari and Di Mascio (2013) applied two approaches for the simulation of propeller turbulent flows, a RANS method and a DES, in order to assess advantages and limits of the two different turbulence models, see Fig. 7.3.2. As far as global quantities are concerned, their study shows that the two methods perform equally well. On the contrary, the comparison between the approaches shows that the RANSE approach dissipates the local flow features very quickly and, when all other parameters are identical to the corresponding DES, this over–dissipation has been caused by the eddy–viscosity modeling. Furthermore, the growth of turbulent viscosity, strictly connected to the velocity gradients, is such that the stronger the tip vortices, the sooner they are dissipated. On the contrary, the DES method allows to capture the tip vortices evolution as long as the mesh is reasonably refined with a good qualitative and quantitative agreement with experiments.

7.3.2 Benchmark cases

INSEAN E779A: The E779A is a four blade propeller of modified type Wageningen, low-skew, with a uniform pitch (P/D = 1.1), a small forward rake and a diameter of 227.2 mm.

In 1997 INSEAN started a project aimed at obtaining high quality propeller flow data for CFD validation. Measurements of velocity fields, radiated pressure fields cavitation patterns were performed. The present database was obtained by using LDV and 2D-PIV techniques. LDV was used to perform the survey of the 3 velocity components in transversal planes, while PIV measurements were performed along longitudinal planes, acquiring the axial and the radial velocity components. All the measurements were performed in phase with propeller angular position. Furthermore recently radiated pressure fluctuation and cavitation pattern has been measured.

Propeller geometry, velocity, pressure fluctuations and cavitation pattern data are available for downloading at http://crm.insean.it/E779A

PPTC: The PPTC (Potsdam Propeller Test Case) was published in the scope of the Propeller Performance Workshop held at the SMP-2011 (Abdel-Maksoud, 2011). The five bladed CP model propeller has a diameter of 250mm and mean pitch of 1.57. The experimental investigation in a cavitation tunnel includes open water test, LDV velocity field measurements at several planes and cavitation tests in different operation conditions. The geometry and test results are published and open to the public. (Barkmann et al., 2011)

The Propulsion Committee of ITTC is using PPTC as benchmark case for scaling of conventional propeller open water data.

P5168: P5168 is a five-bladed CPP propeller with a design advance ratio of 1.27 and diameter of 402.7 mm. The measurements were made in The David Taylor Variable Pressure Water Tunnel using LDV system. The measurements were made in order to examine the behaviour of the tip-vortex flow. All velocity components have been resolved in the rotating frame of the propeller with sufficient spatial resolution to reveal detailed flow. In addition full Reynolds stress tensor was measured for the primary advance coefficient (Chesnakas, 1998).

Waterjets: AxWJ-2 Waterjet Pump. An axial flow waterjet pump (AxWJ-2) has been designed, fabricated, and tested by researchers from Johns Hopkins University and NSWCCD. Measurements of the total head rise and shaft torque on flow through the pump have been taken at a range of flow conditions through cavitation breakdown is reported in (Chesnakas et al., 2009). These results have been used for CFD validation in some research studies.

7.3.3 V&V procedures and needs

For propeller related problems V&V is usually performed considering global quantities such as thrust and torque coefficient. An example of the V&V process can be found from Chase and Carrica (2012). The verification study was performed for one advance coefficient on four grids coarseness and three time step sizes. The study shows that grid refinement has a weak effect on thrust and torque but very strong affects on the wake details.

7.4 Seakeeping

The development and the use of numerical methods for seakeeping related problems continued at high level over the past three years. Indeed, it appears that current CFD tools are mature to provide accurate predictions in seakeeping related problems. The number of publications and submissions to workshops (such as the Gothenburg CFD workshop 2010) is increasing. However, it is still true that two primary issues limit the widespread use of these methods:

- CFD application requires significant CPU time; this is particularly true for seakeeping problems, for which the required simulation times are generally significantly larger than resistance and many manoeuvring problems. Therefore, CFD approaches are not efficient methods for obtaining the Response Amplitude Operators (RAOs) for a range of wave headings, frequencies and wave steepness; similar problems arise when dealing with irregular waves.
- CFD methods are still relatively poor at simulating the disturbed ship waves in the far field domain.

Concerning the first issue, the problem can be partially alleviated by using the procedure proposed by Mousaviraad et al. (2010), that computes the RAOs for one Froude number in a single run, thus reducing significantly the computational time. The methodology has been proven to be as accurate as the standard single wave run, at least within the hypothesis of linear response (small steepness of the incoming waves).

Nevertheless, use of CFD simulations for analysis of seakeeping performance of surface vehicles is becoming common, as indicated by the research work available in the literature and presented at the most important symposiums in ship hydrodynamics.

7.4.1 Review of recent literature

In the last CFD workshop held in Gothenburg in 2010 (Larsson et al. 2014), several seakeeping test cases were included. Namely, data was made available for three hull forms (KVLCC2, KCS and DTMB5415) under different conditions: forward speed diffraction and roll decay for the DTMB5415, prediction of motions and wave resistance, including free to surge and restricted for KCS and KVLCC2. There were four contributions from several institutions for the forward speed diffraction test case, using different CFD codes (CFDShip-Iowa, ICARE, SURF and FLUENT); four contributions for the roll decay case (CFDShip-Iowa, COMET, ICARE and FLUENT); five contributions for heave and pitch of KCS (CFDShip-Iowa, COMET, FreSco+, WISDAM); five contributions for KVLCC2 in head waves free or restrained surge (ISIS, ICARE, COMET, CFDShip-Iowa and RIAM-CMEM). Results have been deeply analyzed in Larsson et al. (2014). The analysis included comparisons in terms of motions and forces, as well as wave pattern and flow field at the nominal wake plane (only for the forward speed diffraction case). It was highlighted a total average error of about 25%, equally distributed among geometries/test cases. Larger errors have been observed in the prediction of the wave resistance; however, as reported in the conclusions, efforts should be also devoted to a correct estimation of experimental uncertainty including facility bias.

Carrica et al. (2011) presented computations of the KCS model in head waves, i.e. one of the cases proposed at the Gothenburg 2010 CFD Workshop. Results are compared with other submission to the workshop and with experimental data; in particular, comparisons of zeroth and first harmonic (amplitudes and phase) for both total resistance coefficient and motions (heave and pitch) are included. The conclusions are similar to those reported in Larsson et al. (2014), i.e. it is observed that pitch and heave are much better predicted than resistance.

The same case has been investigated also by Simonsen et al. (2013), where a complementary EFD and CFD analysis is performed. The EFD data was made available for the Gothenburg 2010 Workshop. CFD analysis was conducted using two URANS code (CFDShip-Iowa and Star-CCM+) and a potential flow tool (AEGIR). In order to investigate ship response under resonance and maximum exciting conditions, the study was pursued for three speeds and several wave conditions. It has been found that the ship responds strongly when the resonance and maximum exciting conditions are met; this allowed the identification of the speed for maximum response. A detailed flow field analysis was conducted, highlighting a very complex and time-varying flow pattern. Similar conclusions as drawn at the Gothenburg workshop were reported. For integral quantities the numerical/experimental comparison shows better agreement in calm water than in waves. Larger errors were observed for resistance in waves. In particular, the mean resistance is well predicted, whereas the first harmonic amplitude is largely under-predicted. Comparing CFD with potential theory results, it was observed that URANS codes are in closer agreement with the experiments compared to inviscid predictions.

The Froude number for maximum response, resonance conditions and wave excitation forces were the main topics in the research by Sadat-Hosseini et al. (2013). In this paper CFD simulations were used to compute added resistance and motions of the KVLCC2 model at two speeds. Validation was performed against experimental data at the lower speed. Local velocity flow field is compared with PIV measurements conducted by Osaka University.

In Guo et al. (2012) results for the KVLCC2 in head wave have been reported; the paper summarizes tests presented at the Gothenburg 2010 workshop using the code ISIS. A comprehensive validation and verification work demonstrates that reliable numerical results can be obtained both in calm water and in head waves. In order to investigate the contribution to added resistance from ship motions, the analysis has been pursued in both free to heave and pitch and fixed conditions; results show that ship motion-induced added resistance is negligible when the wavelength is small enough ($\lambda < 0.6\, L_{pp}$). Moreover, it has been highlighted that the pitch and heave motions in regular head waves can be estimated accurately by both CFD and strip theory, whereas, added resistance predicted by CFD simulations are in better agreement than those based on strip theory. Added resistance and motions have been computed by Seo et al. (2013) using a non-viscous Cartesian grid method; results were compared with predictions from strip theory and Rankine panel method as well as experimental data. Tests have been conducted for three ship geometries. Fairly good agreement was obtained for all methods, including the Cartesian Euler-based methodology.

A CFD analysis of the seakeeping performance for a fast catamaran has been conducted by Castiglione et al. (2010). Several conditions were tested and documented. Comparisons against experimental tests conducted at Delft University were performed. Fairly good agreement has been reported, indicating that CFD based tools can also be employed for the analysis of behavior of multi hull vessels in waves. In He et al. (2013) the same model was considered for uncertainty quantification of resistance, motions and slamming loads in variable regular waves representing a given sea state, and compared to irregular waves and deterministic regular wave studies.

7.4.2 Benchmark cases

As it was also concluded by the previous ITTC Specialist Committee on Computational Fluid Dynamics in Marine Hydrodynamics, validation data for seakeeping including motions, hydrodynamic loads, and flow field are still scarce. Some efforts have been made during recent years, especially within working groups (such as the AVT-NATO 216 group) and in the framework of collaborative research projects. Seakeeping data for code validation have been collected for the Delft catamaran in waves at CNR-INSEAN (Broglia et al. 2011, Bouscasse et al. 2013). Seakeeping parameters including motions and time histories of resistance are available from transient tests, regular and irregular waves for a range of speeds, wave lengths, and wave steepness.

Data collected for the latest CFD Workshop Gothenburg 2010 are also available, but, as concluded by the organizing committee, experimental uncertainty analysis, including facility bias, is still an issue.

Another important group is the Cooperative Research Ships (CRS), which includes 23 companies with a common interest in research in ship hydrodynamics. Similarly to one of the test cases proposed for the Gothenburg 2010 CFD Workshop, the CRS organized a workshop in which a number of research groups were invited to carry out seakeeping predictions for a number of ships. For these ships, model tests were carried out in the past and results were made available to workshop participants. Summary of the workshop is reported in Bunnik et al. (2010). Two test cases were considered: a container ship and a ferry, advancing at forward speed in regular waves. The container ship model test data was available to the participants prior to the start of the comparative study. The ferry was a blind test case. Experimental data includes motions and hydrodynamic loads for several conditions (wave length/amplitude, ship speed and wave direction). The comparisons conducted highlighted the superior accuracy provided by CFD codes in those cases where either nonlinear or viscous effects are significant. Moreover, similarly to the Gothenburg 2010 CFD Workshop conclusions, a general under-prediction of the added resistance was revealed. This, once again, highlights the difficulties in producing accurate predictions and measurements of this quantity.

7.4.3 V&V procedures and needs

For seakeeping related problems (i.e. regular head waves), V&V is usually performed considering global quantities such as first harmonic amplitudes and phases of motions and forces. V&V is then conducted for these quantities as for the resistance in calm water (see for example Larsson et al. 2014, Carrica et al. 2011).

7.5 Manoeuvring

The development and the use of numerical methods for manoeuvring related problems continued at steady pace over the past three years. The number of publications which deal with both steady and unsteady RANS-based manoeuvring simulations is increasing, as well as the level of complexity, the geometrical details taken into account (movable appendages, propellers) and the inclusion of other important aspects (such as controllers, influence of the air, occurrence of breaking wave phenomena, air entrapment and so forth). However, nowadays RANS based numerical simulation of ship manoeuvres at model or full scale is still a challenge, due to both the complexity of the physical phenomena involved and the level of capability and resources required to perform computations.

Inviscid techniques are still object of development and, certainly, of ample use. However, they are mainly employed for particular manoeuvring related problems (such as ship-ship interaction, confined water manoeuvring). Therefore, this section will discuss only recent developments and applications for CFD based methods, i.e. steady and unsteady RANS based simulations.

7.5.1 Review of recent literature

Recent applications and development can be divided in simulations of prescribed and free manoeuvres. Prescribed manoeuvres include steady drift and dynamic captive model tests; these simulations are mainly used in lieu of experiments to obtain coefficients to be employed in system-based models to predict actual dynamic manoeuvres. To this class of CFD applications belong, for example, Planar Motion Mechanism (PMM) computations and rotating arm computations, including all conditions in which one or more degrees or freedom are prescribed. However, some degrees of freedom can be free, as in a pure yaw PMM simulation free to roll.

Free running simulations comprise all those cases where the trajectory is one of the unknowns of the problem, as a result, for example, of the actuation of one control surface; in this class of applications all the classic free running manoeuvres (zig-zag, turning circle, spiral, etc.) are included.

The state of the art in CFD development and applications for manoeuvring simulation is summarized in the latest workshops on the topic, namely the SIMMAN 2008 (Stern and Augdrup 2008, Stern et al. 2011) and the Gothenburg 2010 CFD Workshop (Larsson et al. 2014), by the review by Stern et al. (2012), and by the following literature review.

Simulation of Prescribed Manoeuvres

Literature dealing with captive motion computations is extensive. As discussed here, results mostly show that CFD is mature for this kind of application.

In Toxopeus (2011) CFD computations for KLVCC2 in steady drift and steady turn are presented. The aim of the study is to verify and validate the prediction of the influence of the water depth on flow field and forces and moments. Several grids were used to investigate the discretization error. In general, the uncertainties were found to increase with increased flow complexity, i.e. for larger drift angles or yaw rates.

Also for KVLCC2, Toxopeus et al. (2013) reported the activity of the NATO AVT-161 working group; similar test cases to those reported in the previous paper were considered. Comparisons between predictions by different research groups using several CFD codes (CFDShip-Iowa, ReFreSco, STAR-CCM+ and ISIS-CFD) were reported. Detailed verification and validation studies of the solutions were conducted, revealing that relatively fine grids are required to keep uncertainties within reasonable levels, unless wall functions are used. Moreover, validation of the flow field shows that turbulence modeling plays an important role, especially in the prediction of the wake of the ship. More advanced turbulence models such as EASM or ARS-DES produce wake fields with better resolution of the hook shape found in the experimental results. Validation of global quantities (such as resistance, lateral force and yaw moment) shows low errors and uncertainty levels for the straight ahead deep water condition, whereas higher numerical uncertainty is seen the cases with drift, steady turn and in shallow water. This suggests that modeling errors still persist (experimental conditions, presence of side walls). The authors also stress the need of well documented and validated low uncertainty experimental tests.

Steady drift and steady turn computations, but for underwater vehicles were pursued by Druet et al. (2011), Kim et al. (2012b) and Delanay (2011); also for submarines, larger deviation/uncertainty is observed for large drift angle or yaw rate. Steady turn computations for a twin screw ship was studied numerically and experimentally by Mauro et al. (2012), where CFD was applied analyze propeller overload and unbalance during a tight manoeuvre.

Static drift computations were performed for a fast semi-displacement catamaran by Visonneau et al. (2012); results show good agreement with experiments, including prediction of air entrapment from bow breaking waves. It is worth to note the use of an automatic mesh refinement (AMR) approach coupled with a sliding grid approach developed by the authors. The use of AMR has proven to increase the accuracy of the results. The capabilities of the methodology were demonstrated for several cases, including a ventilated propeller and a self-propelled KCS in straight ahead motion.

Particular interest has been devoted to the analysis of a vessel advancing at very large drift angles, were the simulation of massively separated flows is a challenging problem. Indeed, correct prediction of the onset and progression of the large vortical structures (see Fig. 7.5.1) shed from the hull and generated from the wave breaking and the following splash up phenomena requires not only a very fine grids but also sophisticated turbulence models, as it has been shown in Pinto-Heredero et al. (2010) and Xing et al. (2012). As in Xing et al. (2012), Ismail et al. (2010) studied KVLCC2 in steady drift conditions, focusing on evaluation of linear and nonlinear convection schemes on non-orthogonal grids.

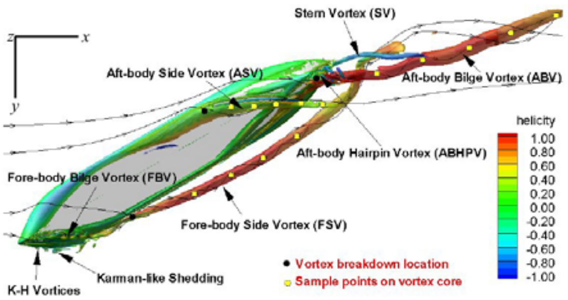


Figure 7.5.1. Vortical structures around KVLCC2 in steady drift (Xing et al. 2012).

Dynamic tests have also become rather common. However, due to the need of large computational resources and special techniques in order to deal with body motions (use of non-inertial frames of reference, dynamic overset grids, 6DoF capabilities in a free surface environment, if any degree of freedom is left free) the literature is still relatively scarce.

Sakamoto et al. (2012a,b) report a complete PMM program using the unsteady RANS solver CFDShip-IOWA. The ship considered is the surface combatant model DTMB 5415 in bare hull configuration. The paper is divided in two parts, the first reporting verification and validation of force and moment coefficients, hydrodynamic derivatives, and reconstruction of force and moment coefficients from resultant hydrodynamic derivatives. In the second part, verification and validation of local flow quantities is presented. The main conclusion, similar to those from SIMMAN 2008, was that CFD methods are mature enough to simulate static and dynamic captive model tests, and obtain derivatives needed by system-based methods to simulate manoeuvres.

PMM for the ONR tumblehome was investigated by Mousaviraad et al. (2012), where complementary EFD and CFD were performed for the analysis of head wind effects on the resistance during straight ahead test and on the hydrodynamic loads during experimental and virtual PMM tests. It has been observed that, for static and dynamic PMM. the wind effects are significant when the model experiences large drift angles relative to the wind. The largest wind effects are observed for yaw & drift followed by pure yaw, while for pure sway the wind effects are minimal due to the near zero wind drift angle.

A similar analysis was presented in Di Mascio et al. (2011) for a twin-screw single rudder tanker-like ship. Results were used to obtain coefficients for a system-based mathematical model, which was then used to predict zig-zag and turning circle manoeuvres of the vessel. Also in Simonsen et al. (2012), some of the coefficients needed by a system-based model were computed from static CFD computations.

It is concluded that, in most cases, CFD methods are a suitable alternative to model experiments for static and dynamic captive model tests.

Simulation of Free Manoeuvres.

Direct prediction of ship manoeuvres with CFD has become possible and its use is increasing, though still mostly in a research context. There are many issues that make this kind of computations challenging. Indeed, direct manoeuvring prediction of surface ships or an underwater vehicles requires the use of special techniques in order to deal with moving appendages (such as dynamic overset grids, sliding grids or re-meshing), full 6DoF and self-propulsion capabilities. Free running CFD computations are also very demanding on computational resources, being presently possible only on high-performance computing environments. However, some applications have been recently presented.

In Broglia et al. (2011a) and Dubbioso et al. (2012a,b) a turning circle manoeuvre of a twin screw tanker vessel has been studied. The vessel was equipped with two different stern appendage configurations, with single or twin rudders and altered skeg arrangements, that experimental trials show to have a strong influence on the turning ability of the vessel. CFD computations were able to predict the turning manoeuvre for both configurations with satisfactory agreement with free running tests. The authors perform full 6DoF computations in a free surface environment including moving rudders, while the propellers were modeled using body forces. Durante et al. (2010) show that accurate prediction of manoeuvres requires inclusion of the lateral propeller forces. In the cited papers, the propeller model reported in Broglia et al. (2012) was used.

Simulations of captive and free manoeuvres of a submarine were presented by Chase et al. (2013). In this work the use of dynamic overset grids allowed direct simulation of the rotating propeller, which was alternatively modeled using the propeller code PUF-14. Results are presented for several manoeuvers, including a free running 20/10 horizontal overshoot manoeuvre. In Fig. 7.5.2 a view of the surface pressure and the vortical structures shed from the propeller is shown.

Zigzag manoeuvres of KCS with direct representation of moving rudder and propeller were performed by Mofidi and Carrica (2014), see Fig. 7.5.3. The authors computed 10/10 and 15/1 manoeuvres, and compared extensively against experimental data for pitch, roll, yaw, yaw rate, propeller thrust and torque, and ship speed. They also analyzed the forces and moments on the rudder, studying separation and propeller-rudder interaction during the manoeuver.

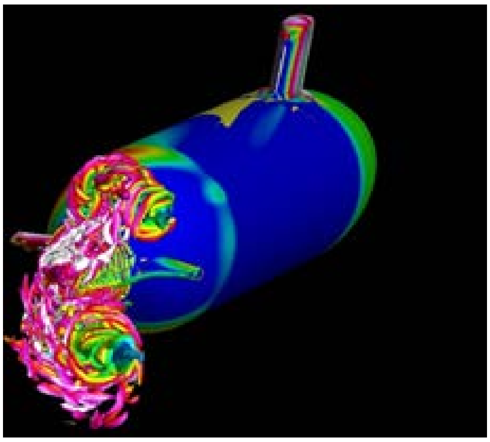


Figure 7.5.2. Horizontal overshoot manoeuvre with discretized propeller (Chase et al. 2013).

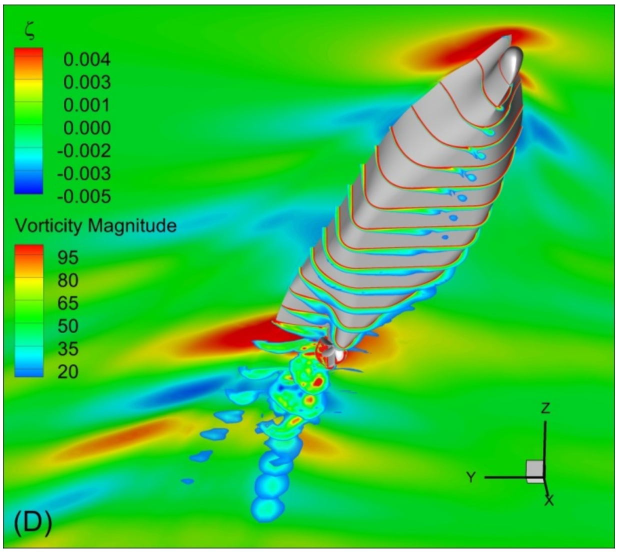


Figure 7.5.3. Instantaneous view of zigzag manoeuvre of KCS with discretized propeller (Mofidi et al. 2014).

Manoeuvring in waves

RANS simulations of manoeuvring in waves have also become possible. In Carrica et al. (2013) unsteady RANS computations of standard manoeuvres in both calm water and in waves are performed for a surface combatant at model and full scale. Two types of manoeuvres are simulated in calm water: steady turn and zigzag, and a turning circle is considered in waves. Some calm water computations were validated against experimental data, showing comparisons for time histories of kinematical and dynamic quantities, as well as for integral variables. Differences between CFD and experiments were found to be mostly within 10%, which can be considered highly satisfactory given the degree of complexity of these computations. Of major interest in this paper is the study conducted for the turning ability characteristics of the vessel when manoeuvring in wave (Fig. 7.5.4 shows an overview of the solution during the manoeuvre). The study has shown that when the ship is manoeuvring in waves (head waves during the approaching phase) the trajectory becomes elliptical, due to the difference in the ship velocity in head and following waves.

Dynamic stability problems can be classified as mixed seakeeping/manoeuvers in waves. Carrica et al. (2012b) focused on the mechanisms of broaching in following regular waves. The vessel considered is the fully appended ONR Tumblehome model, including bilge keels, skeg, shafts, struts and rudders. The movement of the rudders (used to control the headings) is controlled by means of proportional and proportional–integral autopilots. The propeller is directly gridded. Results were validated against experiments of an auto-piloted, self-propelled model ship, the agreement was rather satisfactory, taking also in consideration the complexity of the study. The flow field and forces and moments on the hull and individual appendages were deeply analyzed, allowing identification of the mechanisms leading to the broaching event. It was found that several reasons contributed to the occurrence of a broaching event, but the use of a slightly better autopilot prevents broaching under identical operating conditions. A similar analysis was conducted by Sadat-Hosseini *et al.* (2011) who used a body force propeller model and focused on comparing EFD data, CFD results and system based simulations.

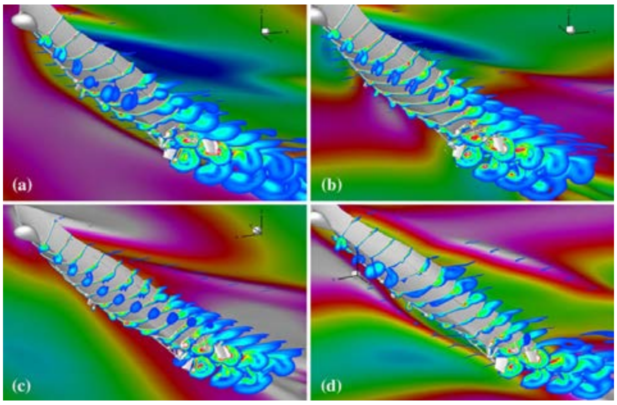


Figure 7.5.4. Free surface and axial velocity contour during a steady turn in waves for the surface combatant DTMB 5415 (Carrica et al. 2013).

Manoeuvring in calm water and in waves for an SES vessel was investigated by Mousaviraad et al. (2012). The simulations take into consideration the waterjet propulsors, including nozzles and reverse buckets. Shallow water and wave effects were studied for turning circles and zig-zag manoeuvres. The cushion pressure was modeled (see Bushan et al. 2011).

Finally, it is worth mentioning the use of CFD simulations for derivation of manoeuvring coefficient needed in system based mathematical models. In Araki et al. (2012) system identification techniques are used to predict the manoeuvring coefficients from several EFD, systems based and also CFD free-running trials. Notably, CFD gives not only the ship motions but also the total and component hydrodynamic forces/moments during the free-running simulations, which is helpful for estimating the manoeuvring coefficients. Several types of free-running (turning circle, zigzag and large angle zigzag) tests were considered, using both EFD and CFD, to examine which free-running trial gives the best manoeuvring coefficients using system identification. It has been shown that the set of manoeuvring coefficients estimated by the constrained least square method using combined CFD free-running trial data show the most generalized results covering a wide range of manoeuvres.

7.5.2 Benchmark cases

Some efforts to collect manoeuvring benchmark data are of note in recent years. However, as already concluded by the previous ITTC CFD committee, the main contribution to benchmark data is still connected to the Workshop on Verification and Validation of Ship Manoeuvring Simulation Methods, SIMMAN 2008 (Stern et al. 2011). New benchmark data will be released for the next workshop, SIMMAN 2014, which will be held in December 2014 in Denmark.

Sanada et al. (2012, 2013) collected experimental data for the ONR Tumblehome manoeuvring in calm water and in waves. The measurements were performed for several manoeuvres and with comprehensive repeated tests, making this experimental work a valuable database for CFD benchmarking.

Availability of velocity measurements is of paramount importance for CFD validation. Indeed, these data would allow checks on local quantities (such as mean velocity, Reynolds stresses and turbulent kinetic energy). In this field some progress has been recently made, mainly using optical measurement techniques such as Particle Image Velocimetry (PIV). Flow field measurements are available mainly for captive model tests; Yoon (2009) provides stereo PIV measurements of the velocity fields on cross plane around the DTMB5415 undergoing PMM motions. Measurements around a similar surface combatant model in steady turning have been collected by Atsavapranee et al. (2010), whereas, Irvine et al. (2013) performed forces measurements as well as local flow PIV acquisition in forward speed roll decay motion in calm water.

Flow field data was collected in the framework of the activity of AVT-NATO 183 research group; in particular, stereo PIV measurements around the Delft catamaran advancing in steady drift were performed at CNR-INSEAN (Broglia et al. 2011b). Velocity measurements were collected for two Froude number and two drift angles (6° and 9° degrees). Also, as part of the AVT-NATO group, tomographic PIV data are being collected around the DTMB5415 in straight ahead and in steady drift (a large degree angle) conditions (Egeberg et al. 2014).

7.5.3 V&V procedures and needs

Little to no literature is available related to V&V procedures for manoeuvring. Verification is challenging due to difficulties to define proper measures and to achieve at least three grids in the asymptotic range. Validation (comparison with experimental data) assessment is provided for global quantities such as the trajectory parameters (transfer, advancement, tactical and turning diameters, overshoot angle, etc.), dynamical parameters (speed loss, yaw rate, drift angles, etc.) or by comparison with the trajectory, see for example Carrica et al. (2013) and Broglia et al. (2011).

7.6 Ocean Engineering

7.6.1 Review of recent literature

In the field of ocean engineering, except for internal flow problems, CFD use has been limited to applications for which viscous flow effects are not negligible: free-surface flow around offshore structures, ocean renewable energy, and fluid-structure interaction of ocean structures. Although CFD applications to the problems are increasingly popular these days, there is still a lot of room for improvement, which requires extensive amount of research.

There is literature that deals with free-surface flows around offshore structures. Problems of interest vary from green water shipping to tidal stream energy conversion systems.

FPSO's are generally operated in a specific region and positioned to meet mostly head or bow waves in order to reduce roll motions. In Lim et al. (2012), experimental results for three different FPSO bow shapes in regular head waves were analysed and compared to each other. Also CFD computations were carried out as a sample validation case for the database built for CFD code validation.

Nielsen and Mayer (2004) investigated two cases. First, green water on a fixed vessel was analysed, where resulting water height on deck, and impact pressure on a deck mounted structure were computed. Second, a full green water incident including vessel motions was modelled. In these computations, the vertical motion was modelled by the use of transfer functions for heave and pitch, but the rotational contribution from the pitch motion was neglected. The computed water height on deck was compared to experimental data.

Buchner et al. (2001) developed a numerical time domain simulation model for prediction of the hydrodynamic response of an LNG FPSO with an alongside moored LNG carrier. The situation with two floating bodies in close proximity resulted in a strong and complex hydrodynamic interaction. Their use of a free surface lid in the multiple-body diffraction analysis resulted in an important improvement of drift force prediction and resulting relative sway and yaw motions.

In Kim (2011), a two-dimensional floating body with a moon pool under forced heave motion, including a piston mode, was numerically simulated. A dynamic CFD simulation was carried out to thoroughly investigate the flow field around a two-dimensional moon pool over various heaving frequencies. The effects of vortex shedding and viscosity were investigated by changing the corner shapes of the floating body and solving the Euler equations. The flow fields, including the velocity, vorticity, and pressure fields, were analyzed to understand and determine the mechanisms of wave elevation, damping, and sway force.

Park et al. (2007) carried out numerical analysis of a moon pool in rough seas. From hydrodynamic viewpoint, a moon pool of drill ships can cause various problems. Among them, there are two major problems such as increased resistance and overflow on the deck due to pumping up phenomena. To overcome these inherent problems, various numerical analyses to find optimum moon pool shapes were conducted.

In dealing with VIV problems of risers, structures are conveniently described by Lagrangian formulations, while fluids are usually described by Eulerian formulations. In other words, FEM is used for the structure solver, while methods like FVM, FDM, and

FEM are utilized for the fluid solver in the ALE formulation. The coupling requires tight integration of those two solvers. Chen and Kim (2012) successfully applied such a method and presented the results of the dynamic effect of internal flow considering FSI for a marine riser in an external shear current.

In Halkyard et al. (2006), helical strakes were employed to mitigate VIM. The paper reported on the results of benchmarking studies that had been conducted to compare model tests with CFD. The paper discusses comparisons of CFD with model tests, "best practices" for the use of CFD for these classes of problems and issues related to turbulence modelling and meshing of problems at large Reynolds numbers.

Two computational procedures, based on the blade element momentum theory and CFD, were developed for open water performance prediction of horizontal axis tidal stream turbines (Lee et al. 2012). The developed procedures were verified by comparison with other computational results and existing experimental data and then, applied to a turbine design process (see Fig. 7.6.1). Malki et al. (2013) presented a coupling method of the blade element momentum and CFD, applied to a horizontal tidal stream turbine.

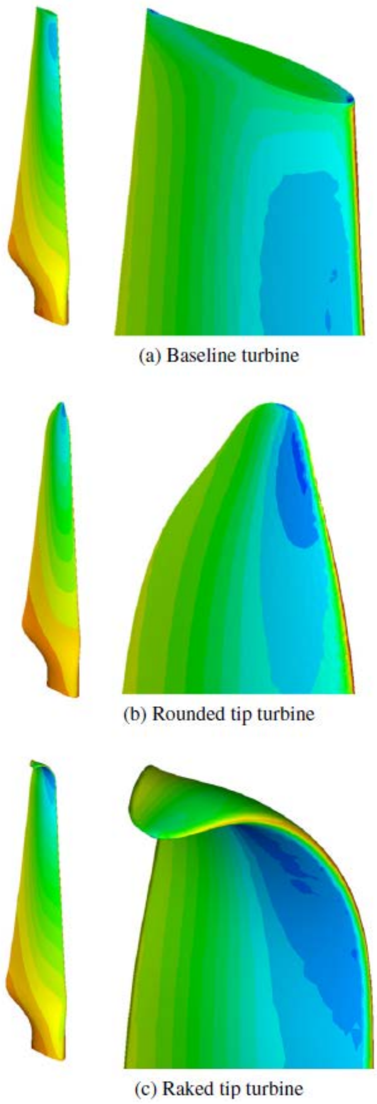


Figure 7.6.1. Contours of pressure coefficient on the blade surface of tidal stream turbines (Lee et al. 2012) .

A numerical study of the Savonius type direct drive turbine incorporated in the rear bottom of typical chamber geometry of an oscillating water column chamber (OWC) for wave energy conversion was presented in Zullah and Lee (2013). The study dealt with a numerical modelling devoted to investigate the effect of wave on the performance and internal flow of the Savonius turbine in the components of an oscillating water column (OWC) system used for the wave energy capture. Another application to wave energy system is Yu and Li (2013). They studied the hydrodynamics performance of a floating-point absorber wave energy system and found that nonlinear effects, including wave-overtopping induced forces, are significant (see Figure 7.6.2).

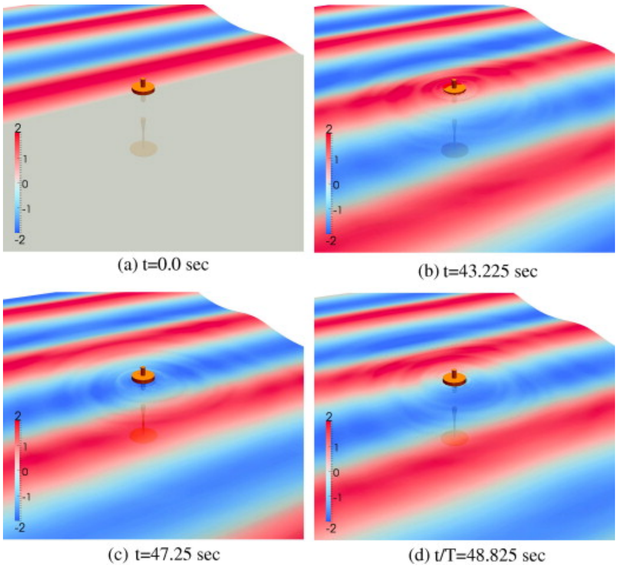


Figure 7.6.2. Free-surface elevation around a two-body floating-pint absorber (Yu and Li 2013).

Finnegan and Goggins (2012) described a CFD method to generate linear waves in a numerical wave tank. Wave–structure interaction on a floating cylinder was modelled using the method.

7.6.2 Benchmark cases

For the wave run-up problem around an offshore plant-related structure, the most widely accepted benchmark is the experimental data adopted by the 27th ITTC Ocean Engineering Committee. The problem of interest was the free-surface behaviour around and pressure acting on a single truncated circular cylinder in waves. For various wave conditions, wave elevation ahead of and behind the cylinder, wave run-up and pressure on the cylinder were measured. For more details refer to the ITTC Ocean Engineering Committee's report on the benchmark study.

One of the first and most accepted benchmark data sets for horizontal axis tidal stream turbines is provided by Bahaj et al. (2007). A tidal stream turbine model of the most popular type was towed in a towing tank. The tidal turbine's performance was measured and compared with the one predicted by a theoretical formula.

7.6.3 V&V procedures and needs

V&V for CFD simulations of ocean engineering problems are still controversial. Because of the unsteady nature of the problems, it is difficult to understand how to quantify errors in the more or less random temporal variations. Moreover, full 6DOF motion in response to external environment is susceptible to various types of uncertainties.

8. CONCLUSIONS

In response to the Terms of Reference, this report presents reviews of different CFD applications for marine hydrodynamics with additional discussion on benchmark data for validation. Reviews of specific topics associated with steep and breaking waves, steady and unsteady flow field predictions and new directions of developments are also included. In addition, wake scaling and verification and validation are addressed for practical CFD applications.

9. RECOMMENDATIONS TO THE FULL CONFERENCE

The Specialist Committee on CFD in Marine Hydrodynamics recommends to the Full Conference to

- Adopt the revised guideline 7.5-03-02-03 Practical Guidelines for Ship CFD Applications.

- Adopt the new guideline 7.5-03-02-04 Practical Guidelines for Ship Resistance CFD.

- Adopt the new guideline 7.5-03-03-01 Practical Guidelines for Ship Self-propulsion CFD.

- Adopt the new guideline 7.5-03-03-02 Practical Guidelines for RANS Calculation of Nominal Wakes.

10. REFERENCES

Abdel-Maksoud, M. (Ed.) “Workshop on Cavitation and Propeller Performance”, 2nd Int. Symp. Marine Propulsors, Hamburg, Germany, 2011.

Alin, N., Bensow, R. E., Fureby, C., Huuva, T., Svennberg, U. “Current capabilities of DES and LES for submarines at straight course,” J. Ship Res., Vol. 55, pp. 184-196, 2010.

American Institute of Aeronautics and Astronautics, “AIAA guide for the verification and validation of computational fluid dynamics simulations”, AIAA G-077-1998, 19 pp.

American Society of Mechanical Engineers, “Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer: ASME V&V20”, 30 Jun 2009, 100 pp.

American Society of Mechanical Engineers, “Guide for Verification and Validation in Computational Solid Mechanics: ASME V&V10”, Oct 2006, 36 pp.

Amini, Y., Emdad, H., Farid, M. “A new model to solve fluid-hypo-elastic solid interaction using Smoothed Particle Hydrodynamics.” Euro. J. Mech. B/Fluids, Vol. 30, pp. 184-194, 2011.

Araki, M., Sadat-Hosseini, H., Sanada, Y., Tanimoto, K., Umeda, N., Stern, F. “Estimating maneuvering coefficients using system identification methods with experimental, system-based, and CFD free-running trial data.” Ocean Eng., Vol. 51, pp. 63–84, 2012.

Atsavapranee, P., Miller, R., Dai, C., Klamo, J., Fry, D. “Steady-Turning Experiments and RANS Simulations on a Surface Combatant Hull Form (Model #5617),” 28th Symp. Naval Hydrodyn., Pasadena, CA, 2010.

Bahaj, A.S., Molland, A.F., Chaplin, J.R., Batten, W.M.J. “Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank”, Ren. Energy, Vol. 32, pp. 407–426, 2007.

Bhushan, S., Stern, F., Doctors, L. “Verification and Validation of URANS Wave Resistance for Air Cushion Vehicles, and Comparison With Linear Theory,” J. Ship Res., Vol. 55, pp. 249–267, 2011.

Bhushan, S., Alam, M. F., Walters, D. K. “Evaluation of hybrid RANS/LES models for prediction of flow around surface combatant and Suboff geometries,” Comp. Fluids, Vol. 88, pp. 834-849, 2013.

Bouscasse, B., Broglia R., Stern F. “Experimental investigation of a fast catamaran in head waves”, Ocean Eng., Vol. 72, pp 318-330, 2013.

Barkmann, U., Heinke, H.-J., Lübke L. “Potsdam Propeller Test Case (PPTC). Test Case Description.” 2nd Int. Symp. Marine Propulsors, Hamburg, Germany, 2011.

Broglia, R., Dubbioso, G. and Di Mascio, A. (2011a). "Prediction of Manoeuvring Properties for a Tanker Model by Computational Fluid Dynamics," Proc. AVT PANEL, Specialists Meeting AVT-189/RSM-028 on Assessment of Stability and Control Prediction Methods for NATO Air and Sea Vehicles", Dstl Portsdown West Fareham, Hampshire (UK).

Broglia R., Bouscasse B., Jacob B., Olivieri A., Zaghi S., Stern F. "Calm Water and Sea keeping Investigation for a Fast Catamaran", Proc. 11th Int. Conf. Fast Sea Trans., Honolulu, HI, 2011b.

Broglia, R., Dubbioso, G., Durante, D. Di Mascio, A. "Simulation of turning circle by CFD: Analysis of different propeller models and their effect on manoeuvring prediction." Appl. Ocean Res., Vol. 39, pp. 1–10, 2012.

Broglia, R., Aloisio, G., Falchi, M., Grizzi, S., Zaghi, S., Felli, M., Miozzi, M., Pereira, F., Di Felice, F., Stern, F. "Measurements of the Velocity Field around the DELFT 372 Catamaran in Steady Drift," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Brucker, K., O'Shea, T., Dommermuth, D., Adams, P. "Three-Dimensional Simulations of Deep water Breaking Waves," 28th Symp. Naval Hydrod., Pasadena, CA, 2010.

Buchner, B., et al. Numerical multiple-body simulations of side-by-side mooring to an FPSO. Proc. 11th Int. Symp. Offshore and Polar Eng., 2001.

Bull P.W., 2011, "Predictions of Ship Wakes Using Computational Fluid Dynamics," RINA Proc. Develop. Marine CFD, London, UK.

Bunnik, T., Daalen, E.V., Kapsenberg G., Shin Y., Huijsmans, R., Deng, G., Delhonmeau, G., Kashiwagi, M., Beck, B. "A comparative study on state-of-art prediction tools for seakeeping," 28th Symp. Naval Hydrodyn., Pasadena, CA, 2010.

Cao, H.J., Wan, D.C. "Development of Multidirectional Nonlinear Numerical Wave Tank by naoe-FOAM-SJTU Solver, " Int. J. Ocean System Eng., Vol. 4, pp. 52-59, 2014.

Carrica, P.M., Fu, H., Stern, F. "Computations of self-propulsion free to sink and trim and of motions in head waves of the KRISO Container Ship (KCS) model." Appl. Ocean Res., Vol. 33, pp. 309–320, 2011.

Carrica, P.M., Castro, A., Li, J., Politano, M., Hyman, M. "Towards an air entrainment model," 29th Symp. Naval Hydrod., Gothenburg, Sweden, 2012a.

Carrica, P.M., Sadat-Hosseini, H., Stern, F. "CFD analysis of broaching for a model surface combatant with explicit simulation of moving rudders and rotating propellers." Comput. Fluids, Vol. 53, pp. 117–132, 2012b.

Carrica, P.M., Ismail, F., Hyman, M., Bhushan, S., Stern, F. "Turn and zigzag maneuvers of a surface combatant using a URANS approach with dynamic overset grids," J. Mar. Sci. Technol., Vol. 18, pp. 166–181, 2013.

Castiglione, T., Stern, F., Bova, S., Kandasamy, M. "Numerical investigation of the seakeeping behavior of a catamaran advancing in regular head waves." Ocean Eng., Vol. 38, pp. 1806–1822, 2011.

Castro, A., Carrica, P.M., Stern, F. "Full scale self-propulsion computations using discretized propeller for KRISO containership KCS," Comput. Fluids, Vol. 51, pp. 35–47, 2011.

Chase, N., Carrica, P.M. "Submarine propeller computations and application to self-propulsion of DARPA Suboff." Ocean Eng., Vol. 60, pp. 68-80, 2012.

Chase, N., Michael, T., Carrica, P.M. "Overset simulations of a Submarine in Towed, Self-Propelled and Maneuvering Conditions," Int. Shipbuilding Prog., Vol. 60, pp. 171-205, 2013.

Chen, Z.S., Kim, W.J. "Effect of bidirectional internal flow on fluid-structure interaction dynamics of conveying marine riser model subject to shear current," Int. J. Naval Arch. Ocean Eng., Vol. 4, 2012, pp. 57-70.

Chesnakas, C., Jessup, S. "Experimental characterization of propeller tip flow," 22nd Symp. Naval Hydrodyn., Washington, DC. 1998.

Chesnakas, C. J., Donnelly, M. J., Pfitsch, D.W., Becnel, A. J., Schroeder S. D., "Performance evaluation of the ONR axial waterjet 2 (AxWJ-2)," Hydromech. Dept. Report 2009(089), Carderock Division, NSWC, Bethesda, MD, 2009.

Choi, G-H., Chang, B-J., Hur, J. Cho, D-S. "Comparison of Cavitation Patterns between Model Scale Observations using Model and Full-Scale Wakes and Full Scale Observations for a Propeller of Crude Oil Carrier", J. Soc. Naval Arch. Korea, Vol. 48, 2011.

Christ, D. "The NavalHydro-Pack and its Application in the Simulation of Ship Hydrodynamics," 8th Int. OpenFOAM Workshop, Jeju, Korea, 2013.

Dashtimanesh, A., Ghadimi, P. "A three-dimensional SPH model for detailed study of free surface deformation, just behind a rectangular planing hull," J. Brazilian Soc. Mech. Sci. Eng., Vol. 35, pp. 369-380, 2013.

Day, W.G., Reed A., Hurwitz R.B., "Full-scale propeller disk wake survey and boundary layer velocity profile measurements on the 154-foot ship R/V Athena", DTNSRDC report SPD-0833-01, 1980.

Delaney, K.P. "Computational Investigation Of Non-Body-Of revolution Hull Form Maneuvering Characteristics," 4th Int. Conf. Comput. Meth. Marine Eng., Lisbon, Portugal, 2011.

Di Mascio, A., Broglia, R., Muscari, R., A. Pascarelli, "Nominal Wake Computation", EUCLID RTP 10.12 - Task N1.IN2, 2001.

Di Mascio, A., Dubbioso, G., Notaro, C., Viviani, M. "Investigation of Twin-Screw Naval Ships Maneuverability Behavior." J. Ship Res., Vol. 55, pp. 221–248, 2011.

Dommermuth, D.G., Rhymes, L., Rottman, J.W. "Direct Simulations of Breaking Ocean Waves with Data Assimilation," Oceans '13, San Diego, CA, 2013.

Drouet, A., Reliquet, G., Bardin, A., Jacquin, E., Gentaz, L., Alessandrini, B. "Simulation Of Submarine Manoeuvring Using Navier-Stokes Solver," 4th Int. Conf. Comput. Meth. Marine Eng., Lisbon, Portugal, 2011.

Dubbioso, G., Durante, D., Broglia, R., Di Mascio, A. "CFD Maneuvering Prediction of a Twin Screw Vessel with Different Stern Appendages Configuration." 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012a.

Dubbioso, G., Durante, D., Broglia, R., Mauro, S. "Comparison of experimental and CFD results for a tanker-like vessel." MARSIM 2012, Singapore, 2012b.

Durante, D., Broglia, R., Muscari, R., Di Mascio, A. "Numerical Simulations of a turning circle manoeuvre for a fully appended hull," 28th Symp. Naval Hydrodyn., Pasadena, CA, 2010.

Eça, L., Hoekstra, M., "On the Influence of the Iterative Error in the Numerical Uncertainty of Ship Viscous Flow Calculations", 26th Symp. Naval Hydrodin., Rome, Italy, 2006.

Eça, L., Vaz, G., Hoekstra, M., "Code Verification, Solution Verification and Validation in RANS Solvers", 29th OMAE, Shanghai, China, 2010a.

Eça, L., Vaz, G., Hoekstra, M., "A Verification and Validation Exercise for the Flow Over a Backward Facing Step", 5th Eur. Conf. CFD, ECCOMAS CFD, Eds. Pereira J.C.F., Sequeira A., Lisbon, 2010b.

Eça, L., Hoekstra, M., "Verification and Validation for Marine Application of CFD", 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Eça, L., Vaz, G., Hoekstra, M., "A Procedure for the Estimation of the Numerical Uncertainty of CFD Calculations Based on Grid Refinement studies", J Comput. Phys., Vol. 262, pp. 104-130, 2014.

Egeberg, T.F., Yoon, H., Stern, F., Pettersen, B., Bhushan, S. "3D measurements of vortex shedding from a ship hull by means of tomographic PIV," OMAE2014, San Francisco, CA, 2014.

Finnegan, W., Goggins, J. "Numerical simulation of linear water waves and wave–structure interaction," Ocean Eng., Vol. 43, pp. 23-31, 2012.

Gaggero, S., Villa, D., Viviani, M. Rizzuto, E. "Ship wake scaling and effect on propeller performances", Developments in Maritime Transportation and Exploitation of Sea Resources - 15th Int. Congress Int. Maritime Assoc. Mediterranean, 2013.

Gao, Q., Jin, W., Vassalos, D., The Calculation of Propeller Induced Velocity by RANS and Momentum Theory, J. Marine Sci. Appl., Vol. 11, pp. 164-168, 2012.

Gotoh H., Khayyer A., Ikari H., Tsuruta N. "An Improved 3D Particle Method for Violent Wave Impact Calculations," 2nd Int. Conf. Violent Flows, École Centrale de Nantes, Nantes, France, pp. 188-193, 2012.

Guo, B.J., Steen, S., Deng, G.B. "Seakeeping prediction of KVLCC2 in head waves with RANS." Appl. Ocean Res., Vol. 35, pp. 56–67, 2012.

Haase, M., Iliopulos, F., Davidson, G., Friezer, S., Thomas, G., Binns, J., Davis, M. R. "Application of RANSE-based simulations for resistance prediction of medium-speed catamarans at different scales," 18th Australasian Fluid Mech. Conf., 2012.

Halkyard, J., Atluri, S., Sirnivas, S. "Truss Spar Vortex Induced Motions: Benchmarking of CFD and Model Tests." Proc. 25th OMAE, pp. 883-892, 2006.

He, W., Diez, M., Zoua, Z., Campana, E., Stern, F. "URANS study of Delft catamaran total/added resistance, motions and slamming loads in head sea including irregular wave and uncertainty quantification for variable regular wave and geometry." Ocean Eng., Vol. 74, pp. 189–217, 2013.

Heinke H-J., Hellwig-Rieck K. "Investigation of Scale Effects on Ships with a Wake Equalizing Duct or with Vortex Generator Fins," 2nd Int. Symp. Marine Prop., 2011.

Hoekstra, M., van Terwisga, T., Foeth E.J. "SMP'11 Workshop - Case 1: DelftFoil," 2nd Int. Symp. Marine Propulsors, Hamburg, Germany, 2011.

Huang, B., Wang G., Yu Z., Shi S. "Detached-eddy Simulation for Time-dependent Turbulent Cavitating Flows," J. Mech. Eng., Vol. 25, pp. 484-490, 2012.

INSEAN. "The INSEAN E779A Propeller Dataset." Technical Report INSEAN, 2006.

Irvine, M., Longo, J., Stern, F. "Forward Speed Calm Water Roll Decay for Surface Combatant 5415: Global and Local Flow Measurements," J. Ship Res., Vol. 57, pp. 202-219, 2013.

Ismail, F., Carrica, P.M., Xing, T., Stern, F. "Evaluation of linear and nonlinear convection schemes on multidimensional non-orthogonal grids with applications to KVLCC2 tanker," Int. J. Num. Meth. Fluids, Vol. 64, pp. 850-886, 2010.

ITTC, "Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures", 7.5-03-01-01, 2008, 12 pp.

ITTC, "CFD, General Uncertainty Analysis in CFD Guidelines for RANS Codes", 7.5-03-01-02, 1999, 2 pp.

ITTC, "CFD, General CFD User's Guide", 7.5-03-01-03, 1999, 4 pp.

ITTC, "CFD, General CFD Verification", No. 7.5-03-01-04, 1999, 7 pp.

Jang, H., Mahesh, K. "Large eddy simulation of flow around a reverse rotating propeller," J. Fluid Mech., Vol. 729, pp. 151-279, 2013.

Kagatsume N., Nakagawa M., Mukai N. "Bubble Disappearance Simulation Using Particle Methods," ITE Technical Report, Vol. 35, No.32, pp.13-16, 2011.

Kawamura, T. and Fujisawa, T. "Simulation of Flow around a Self-Propelling Ship Hull using OpenFOAM," 8th Int. OpenFOAM Workshop, Jeju, Korea, 2013.

Khayyer, A., Gotoh, H. "Enhancement of stability and accuracy of the moving particle semi-implicit method," J. Comput. Phys., Vol. 230, pp. 3093-3118, 2011

Kim, M. "CFD Analysis of Two-Dimensional Floating Body with Moon Pool under Forced Heave Motion." J. Korean Soc. Marine Environ. Eng., Vol. 25, No.2, pp. 36-46, 2011.

Kim, K., Leer-Andersen, M., Werner, S., Orych, M. Choi, Y. "Hydrodynamic Optimization of Pre-swirl Stator by CFD and Model Testing", 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012a.

Kim, S.-E., Rhee, S. H., "Assessment of Eight Turbulence Models for a Three-Dimensional Turbulent Boundary Layer Involving Crossflow and Streamwise Vortices," AIAA-Paper 2002-0852, 2002.

Kim, S.E., Rhee, B., Miller, R. "Anatomy of Turbulent Flow around DARPA SUBOFF Body in a Turning Maneuver using High-Fidelity RANS Computations," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012b.

Kinnas, S.A., Jeon, C.H., Tian, Y. "A Hybrid Viscous/Potential Flow Method for the Prediction of the Wetted and Cavitating Performance of Ducted Propellers". SNAME Propellers/Shafting Symp., 2012.

Krasilnikov, V.I. "Self-Propulsion RANS Computations with a Single-Screw Container Ship," 3rd Int. Symp. Marine Propulsors, 2013.

Lam, W.H., Robinson, D.J., Hamill, G.A., Johnston, H.T. "An effective method for comparing the turbulence intensity from LDA measurements and CFD predictions within a ship propeller jet." Ocean Eng., Vol. 52, pp. 105-124, 2012.

Larsson, L., Stern, F., Visonneau, M. Numerical Ship Hydrodynamics - An assessment of the Gothenburg 2010 Workshop. Springer, Netherlands, 2014.

Lee, H.-H., Lim, H.-J., Rhee, S.H. "Experimental Investigation of Green Water on Deck for a CFD Validation Database." Ocean Eng., Vol.42, pp. 47-60, 2012.

Lee, J.H., Park, S., Kim, D.H., S.H. Rhee, S.H., Kim, M.-C. "Computational methods for performance analysis of horizontal axis tidal stream turbines." Appl. Energy, Vol. 98, pp. 512-523, 2012.

Liefvendahl, M., Tröeng, C. "Computation of cycle-to-cycle variation in blade load for a submarine propeller using LES," 2nd Int. Symp. Marine Propulsors, 2011.

Lindau, J.W., Pena, C., Baker, W.J., Dreyer, J.J., Moody, W.L., Kunz, R.F., Paterson, E.G. "Modeling of Cavitating Flow through Waterjet Propulsors." Int. J. Rotating Machinery, Art. 716392, 2012.

Lu, N.X., Svennberg, U., Bark, G., Bensow, R. "Numerical Simulations of the Cavitating Flow on a Marine Propeller". 8th Int. Symp. Cavitation, Singapore, 2012.

Lubin, P., Vincent, S., Abadie, S., Caltagirone, JP. "Three-dimensional Large Eddy Simulation of air entrainment under plunging breaking waves," Coastal Eng., Vol 53, pp. 631-655, 2006.

Lubin, P. and Glockner, S. "Detailed numerical investigation of the three-dimensional flow structures under breaking waves," 7th Conf. Coastal Dynamics, Bordeaux, France, 2013.

Malki, R., Williams, A.J., Croft, T.N., Togneri, M., Masters, I. "A coupled blade element momentum–Computational fluid dynamics model for evaluating tidal stream turbine performance," Appl. Math. Modeling, Vol. 37, pp. 3006-3020, 2013.

Mauro, S., Dubbioso, G., Broglia, R., Muscari, R. "Investigation of Asymmetrical Shaft Power Increase during Ship Maneuvers by means of Model Tests and CFD," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Mofidi, A., Carrica, P.M. "Simulations of zigzag maneuvers for a container ship with direct moving rudder and propeller," Comput. Fluids, Vol. 96, 191-203, 2014.

Morgut, M., Nobile, E. "Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow." Ocean Eng., Vol. 42, pp. 26-34, 2012.

Mousaviraad, M., Carrica, P.M., Huang, J. and Stern, F. "CFD Prediction of Ship Response to Severe Ocean Waves," 27th Symp. Naval Hydrodyn., Seoul, Korea, 2008.

Mousaviraad, M. Carrica, P.M., Stern, F. "Development and validation of harmonic wave group single-run procedure for RAO with comparison to regular wave and transient wave group procedures using URANS," Ocean Eng., Vol. 37, pp. 653-666, 2010.

Mousaviraad, S.M., Bhushan, S., Stern, F. "CFD Prediction of Free-Running SES/ACV Deep and Shallow Water Maneuvering and Course-Keeping in Calm Water and Waves." MARSIM 2012, Singapore, 2012.

Mousaviraad, M., Cook, S., Carrica, P.M., Toda, Y., Stern, F. "Complimentary EFD and CFD on Effects of Headwinds on Towing Tank Resistance and PMM Tests for ONR Tumblehome," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Muscari, R., Di Mascio, A. "Numerical simulation of the flow past a rotating propeller behind a hull," 2nd Int. Symp. Marine Propulsors, 2011.

Muscari, R., Di Mascio, A. "Detached Eddy Simulation of the flow behind an isolated propeller," 3rd Int. Symp. Marine Propulsors, Launceston, Australia, 2013.

Nielsen, K.B., Mayer, S. "Numerical prediction of green water incidents." Ocean Eng., Vol. 31, pp. 363-399, 2004.

Paredes, R., Imas, L. "Application of SPH in Fluid-Structure Interaction Problems Involving Free-surface Hydrodynamics" 11th Int. Conf. Fast Sea Transp., Honolulu, HI, 2011.

Perelman, O., Fournier P., Mace R., Deuff J.B. "DALIDA: Development and validation of tools for hydrodynamic signatures prediction", www.atma.asso.fr, 2012.

Pinkster, J.A., Bhawsinka, K. "A Real-time simulation technique for ship-ship and ship-port interactions," 28th Int. Workshop Water Waves Floating Bodies, L'Isle sur la Aorgue, France, 2013.

Pinto-Heredero, A., Xing, T., Stern, F. "URANS and DES analysis for a Wigley hull at extreme drift angles," J. Mar. Sci. Technol., Vol. 15, pp. 295–315, 2010.

Richardson, L.F., "The approximate arithmetical solution by finite differences of physical problems including differential equations, with an application to the stresses in a masonry dam", Phil. Trans. R. Soc. London, S. A 210, pp. 307–357, 1911.

Rijpkema, D., Starke, B., Bosschers, J. "Numerical Simulation of Propeller-Hull Interaction and Determination of the Effective Wake Field Using a Hybrid RANS-BEM Approach," 3rd Int. Symp. Marine Propulsors, 2013.

Roache, J.R. "Quantification of uncertainty in computational fluid dynamics", Annu Rev Fluid Mech, Vol. 29, pp.123-60, 1997.

Roache, J.R. "Verification and validation in computational science and engineering," Hermosa Publishers, 446 pp., 1998.

Roache, J.R. "Fundamentals of verification and validation," Hermosa Publishers, 476 pp., 2009.

Rosemurgy, W. J., Edmund, D. O., Maki, K. J., Beck, R. F. "A Method for Resistance Prediction in the Design Environment," 11th Int. Conf. Fast Sea Transp., Honolulu, HI, 2011.

Rottman, J., Dommermuth, D., Rhymes, L. "Direct Simulation of Ocean Waves with Data Assimilation," Meeting APS Div. Fluid Dynamics, Pittsburg, PA, 2013.

Rui Z., Chao Y., Li X., Kong, W. "Towards an Entropy-based Detached-Eddy Simulat-ion," Science China Physics, Mechanics and Astronomy, Vol. 56, No. 10, pp. 1970-1980, 2013

Sadat-Hosseini, H., Wu, P.C., Carrica, P.M., Kim, H., Toda, Y., Stern, F. "CFD verification and validation of added resistance and motions of KVLCC2 with fixed and free surge in short and long head waves." Ocean Eng., Vol. 59, pp. 240–273, 2013.

Sadat-Hosseini, H., Carrica, P.M., Stern, F., Umeda, N., Hashimoto, H., Yamamura, S., Mastuda, A. "CFD, system-based and EFD study of ship dynamic in stability events: Surf-riding, periodic motion, and broaching." Ocean Eng., Vol. 38, pp. 88–110, 2011.

Sanada, Y., Tanimoto, K., Takagi, K., Gui, L., Toda, Y., Stern, F. "Trajectories for ONR Tumblehome maneuvering in calm water and waves," Ocean Eng., Vol. 72, pp. 45–65, 2013.

Sanada, Y. Tanimoto, K., Takagi, K., Sano, M., Yeo, D.J., Gui, L., Toda, Y., Stern, F. "Trajectories and Local Flow Field Measurements around ONR Tumblehome in Maneuvering Motion," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012

Sánchez-Caja, A., Martio, J., Saisto I., Siikonen T. "On the enhancement of coupling potential flow models to RANS solvers for the prediction of propeller effective wakes." J. Marine Sci. Technol., DOI 10.1007/s00773-014-0255-4, 2014.

Sakamoto, N., Carrica, P.M., Stern, F. "URANS simulations of static and dynamic maneuvering for surface combatant: part 1. Verification and validation for forces, moment, and hydrodynamic derivatives," J. Mar. Sci. Technol., Vol. 17, pp. 422–445, 2012.

Sakamoto, N., Carrica, P.M., Stern, F. "URANS simulations of static and dynamic maneuvering for surface combatant: part 2. Analysis and validation for local flow characteristics," J. Mar. Sci. Technol., Vol. 17, pp. 446–4468, 2012.

Sasajima H, Tanaka I. "On the Estimation of Wakes of Ships", 11th ITTC Conf., Tokyo, Japan, 1966.

Schuiling, B., Lafeber, F.H., van der Ploeg, A., van Wijngaarden, E. "The Influence of the Wake Scale Effect on the Prediction of Hull Pressures due to Cavitating Propellers", 2nd Int. Symp. Marine Propulsors, 2011.

Sebastien D., "Recent Improvements in the Zonal Detached Eddy Simulation (ZDES) formulation," Theor. Comp. Fluid Dyn., Vol. 26, pp.1-28, 2011.

Seo, M.G., Park, D.M., Yang, K.K., Kim, Y. "Comparative study on computation of ship added resistance in waves." Ocean Eng., Vol. 73, pp. 1–15, 2013.

Shen, Z.R., Wan, D.C. "RANS Computations of Added Resistance and Motions of a Ship in Head Waves," Int. J. Offshore Polar Eng., Vol. 23, No. 4, pp. 263–271, 2013.

Shen, Z.R., Carrica, P.M., Wan, D.C. "Ship Motions of KCS in Head Waves with Rotating Propeller using Overset Grid Method," 33rd Int. Conf. Ocean, Offshore Arctic Eng., San Francisco, CA, 2014.

Simonsen, C., Otzen, J.F., Klimt, C., Larsen, N.L., Stern, F. "Maneuvering Predictions in the Early Design Phase using CFD Generated PMM Data," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Simonsen, C.D., Otzen, J.F., Joncquez, S., Stern, F. "EFD and CFD for KCS heaving and pitching in regular head waves." J. Mar. Sci. Technol., Vol. 18, pp. 435-459, 2013.

Spalart, P.R. "Detached-Eddy Simulation," Ann. Rev. Fluid Mech., Vol. 41, pp. 181-202, 2009.

Starke, B., Windt, J., Raven, H., "Validation of Viscous Flow and Wake Field Predictions for Ships at Full Scale," 26th Symp. Naval Hydrodyn., Rome, Italy, 2006.

Starke, A. R., Bosschers, J. "Analysis of scale effects in ship powering performance using a hybrid RANS-BEM approach," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Stern, F., Agdrup, K., Eds. "Proceedings Workshop on Verification and Validation of Ship Manouvering Simulation Methods (SIMMAN)," Lyngby, Denmark, 2008.

Stern, F., Agdrup, K., Kim, S.Y., Hochbaum, A., Rhee, K.P., Quadvlieg, F., Perdon, P., Hino, T., Broglia, R., Gorski, J. "Experience from SIMMAN 2008—The First Workshop on Verification and Validation of Ship Maneuvering Simulation Methods," J. Ship Res., Vol. 55, pp. 135–147, 2011.

Stern, F., Yang, J., Wang, Z., Sadat-Hosseini, H., Mousaviraad, M., Bhushan, S., Xing, T. "Computational Ship Hydrodynamics: Nowadays and Way Forward," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Tanaka, I. "Scale Effects on Wake Distribution and Viscous Pressure Resistance of Ships". J. Kansai SNAJ, Vol. 146, 1979.

Tavakolinejad, M., Shakeri, M., Duncan J. "Air Bubbles Entrained by Breaking waves Generated by a 2D+T Wave Breaker," 28th Symp. Naval Hydrodyn., Pasadena, CA, 2010.

Toxopeus, S. "Viscous flow calculations For KVLCC2 in deep and shallow water." 4th Int. Conf. Comput. Meth. Marine Eng., Lisbon, Portugal, 2011.

Toxopeus, S., Simonsen, C.D., Guilmineau, E., Visonneau, M., Xing, T., Stern, F. "Investigation of water depth and basin wall effects on KVLCC2 in maneuvering motion using viscous-flow calculations," J. Mar. Sci. Technol., Vol. 18, pp. 471-496, 2013.

van Wijngaarden, E., Bosschers, J., van Terwisga, T. "On Predicting Cavitation-Induced Hull Pressure Fluctuations –Wake

Scale Effects and Signal Variability", IMarEST Ship Noise Vibration Conf., UK, 2010.

Verkuyl, J.B. and Raven H.C. "Joint EFFORT for validation of full-scale viscous-flow predictions", The Naval Architect, Jan., 2003.

Verma, A., Mahesh, K. "A Lagrangian subgrid-scale model with dynamic estimation of Lagrangian time scale for large eddy simulation of complex flows," Phys. Fluids, Vol. 24, pp. 85-98, 2012.

Villa, D., Gaggero, S., Brizzolara, S. "Simulation of ship in self propulsion with different CFD methods: From actuator disk to potential flow/RANS coupled solvers". RINA, - Dev. Marine CFD, 2011.

Viola, I.M., Bot, P., Riotte, M., "On the Uncertainty of CFD in Sail Aerodynamics", Int. J. Num. Meth. Fluids, 2013; 72:1146-1164.

Visonneau, M., Queutey, P., Wackers, J., Deng, G.B., Guilmineau, E. "Sliding Grids and Adaptive Grid Refinement Applied to Ship Hydrodynamics," 29th Symp. Naval Hydrodyn., Gothenburg, Sweden, 2012.

Wan, D.C., Shen, Z.R. "Overset-RANS Computations of Two Surface Ships Moving in Viscous Fluids," Int. J. Comput. Meth., Vol.9, pp. 1-14, 2012.

Wang, J., Wu, Q., Yu H., Xiong, X. "Numerical Simulation of Viscous Wake Field and Resistance Prediction around Slow-Full Ships Considering Free Surface," J. Hydrodyn. Ser A, No..9, 2010. (in Chinese)

Wang, Z., Suh, J., Yang, J. and Stern, F. "Sharp Interface LES of Breaking Waves by an Interface Piercing Body in Orthogonal Curvilinear Coordinates," 50th AIAA Aerosp. Sci. Meeting Nashville, TN, 2012.

Xiao, W., Liu, Y., Wu, G., Yue, D.K.P. "Rogue wave occurrence and dynamics by direct simulations of non-linear wave field evolution," J. Fluid Mech., Vol. 720, pp. 357-392, 2013.

Xing, T., Stern, F., "Factors of Safety for Richardson Extrapolation", J. Fluids Eng., 2010; 132 (6):061403-13.

Xing, T., Bhushan, S., Stern, F. "Vortical and turbulent structures for KVLCC2 at drift angle 0, 12, and 30 degrees," Ocean Eng., Vol. 37, pp. 678–687, 2010.

Yang, H., Kim, B., Yoo, J., Kim, W. "Wake comparison between model and full scale ships using CFD", J. Soc. Naval Architects Korea, Vol. 47, 2010.

Yang, C., Huang, F., Wang, L., Wan, D.C. "Numerical Simulation of Highly Nonlinear Steady and Unsteady Free Surface Flows," J. Hydrodyn., Vol.23, pp. 683-696, 2011.

Yoon, H.Y. "Force/Moment and Phase-Averaged Stereo PIV Flow Measurements for Surface Combatant in PMM Maneuvers," PhD Thesis, The University of Iowa, Iowa City, IA, 2009.

Yu, Y-H., Li, Y. "Reynolds-Averaged Navier–Stokes simulation of the heave performance of a two-body floating-point absorber wave energy system," Comput. Fluids, Vol. 73, pp. 104-114, 2013.

Zhang, Y.X., Wan, D.C. "Numerical Simulation of Green Water Incidents Based on Parallel MPS Method," Proc. 23nd Int. Offshore Polar Eng. Conf., Vol. 3, pp. 931-938, 2013.

Zullah, M.A., Lee, Y.H. "Performance evaluation of a direct drive wave energy converter using CFD." Ren. Energy, Vol. 49, pp. 237-241, 2013.