# The Specialist Committee on Computational Fluid Dynamics

## Final Report and Recommendations to the 26th ITTC

## 1. INTRODUCTION

### 1.1 Membership

*Chairman:*
Dr. Emilio F. Campana
Istituto Nazionale per Studi ed Esperienze di Architettura Navale (INSEAN), ITALY

*Secretary:*
Prof. Takanori Hino
Yokohama National University, JAPAN

*Members:*
Mr. Peter Bull
QinetiQ, UK

Prof. Pablo Carrica
The University of Iowa, USA

Dr. Jin Kim
Maritime & Ocean Engineering Research Institute (MOERI/KORDI), KOREA

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

Dr. Da-Qing Li
SSPA, SWEDEN

Prof. Ning Ma
Shanghai Jiao Tong University, CHINA

Mr. Ilkka Saisto
VTT, Ship Hydrodynamics, FINLAND

Dr. Bram Starke
MARIN, The NETHERLANDS

### 1.2 Meetings

The committee met 4 times:
7-8 Jan 2009, Rome, Italy
8-9 Sept 2009, Iowa City, USA
14-15 June 2010, Gothenburg, Sweden
7,11 December 2010, Gothenburg, Sweden

### 1.3 Tasks

The purpose of this specialist committee is to comprehensively review the past work on the 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 the ITTC. Include resistance, propulsion, propulsors, manoeuvring, seakeeping, ocean engineering and steady and unsteady flow field prediction at model and full scale.

2. Identify CFD elements of importance to the ITTC from a user's point of view, including applicability, accuracy, reliability, time and cost.

3. Review the impact on CFD of different modelling techniques, such as particle methods or Cartesian grid methods.

4. Identify the need for research in the treatment of:

   a. the free surface, unsteady flows, and accurate modelling of turbulence,

   b. surface roughness and the ability to correlate full-scale computations with real ship data.

5. Define which benchmark data are needed for CFD validation. Include the requirement for experimental data.

## 2. QUESTIONNAIRE

The Committee was charged with identifying CFD elements of importance to the ITTC from a user's point of view, including applicability, accuracy, reliability, time and cost.

The Committee prepared a questionnaire on issues regarding usability, difficulty, and applications of CFD in marine hydrodynamics. This addressed five areas: Application, Quality, Simulation Code, Difficulty and Expectations. The questionnaire was circulated by e-mail to all ITTC members and also industries and universities. 194 persons replied to the questionnaire. The Committee believes that the results will be of interest to ITTC members.

### 2.1 User Profile

194 persons replied to the questionnaire, of which 58% belong to an ITTC member institution. 45.6% of the respondents are from Europe, 30.6% from Asia, 19.7% from the USA and the rest from 30 different countries. Figure 2.1 shows the institutional distribution of respondents to the questionnaire. Most answers are from universities and model basins, 34% and 31% respectively. About 20% of respondents (Shipbuilding companies 12% and Engineering for ship design 8%) are from practical users. 71% of respondents are intermediate or advanced users of CFD for ship hydrodynamics applications, see Figure 2.2.

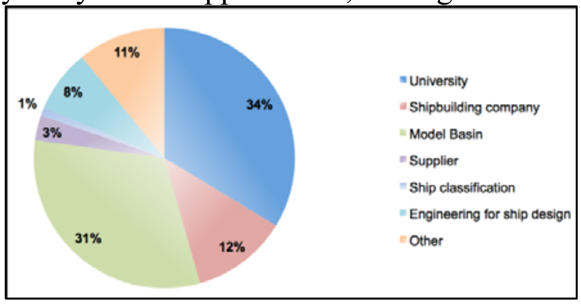



Figure 2.1 Institutional distribution of respondents to the questionnaire

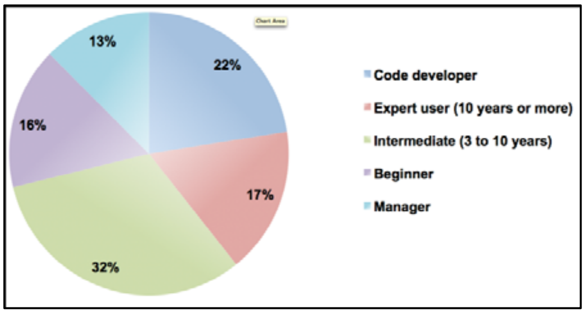



Figure 2.2 Respondents' experience in CFD for marine hydrodynamics

### 2.2 Applications

The questionnaire asked the participants to list their main applications of CFD with multiple choices including resistance, self-propulsion, propulsors, manoeuvring, seakeeping, ocean engineering and others. The most dominant application of CFD is the prediction of resistance (64%), as expected (Figure 2.3). Other applications such as self-propulsion, propulsors, manoeuvring, seakeeping, and ocean engineering are also of great interest with about 40% of respondents applying CFD to these problems.

When asked what areas of resistance prediction are of most interest, prediction of resistance and detailed flow field are viewed as the most important pieces of information extracted from resistance computations (Figure 2.4).

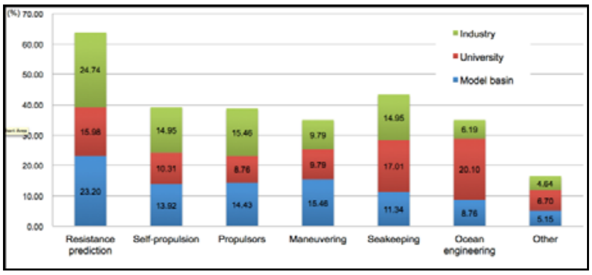



Figure 2.3 Applications of CFD in marine hydrodynamics

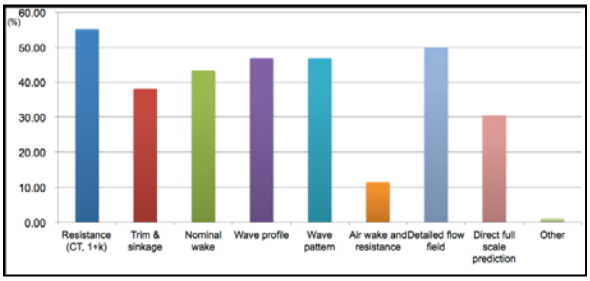



Figure 2.4 Primary interests for resistance prediction with CFD

## 2.3 Quality Check (V&V)

Two of the questions in the questionnaire were related to issues of quality check of the computations. The first question is how often the users check the quality of their computations. As shown in Figure 2.5, most respondents perform quality checks for every computation or often, while a few do it rarely (6%) or sometimes (21%).

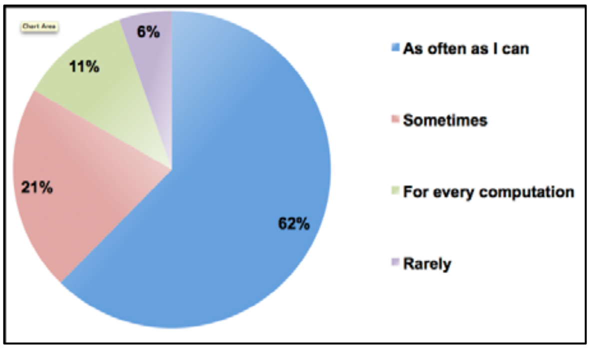



Figure 2.5 Frequency of quality checks of computations

The second question was the method used to check the quality of computations. Unfortunately, only 16% follows the ITTC recommended verification and validation (V&V) procedure 7.5.03.01-01 (ITTC, 2008), see Figure 2.6. However, most respondents are using other V&V procedure (26%) or best practices (23%)

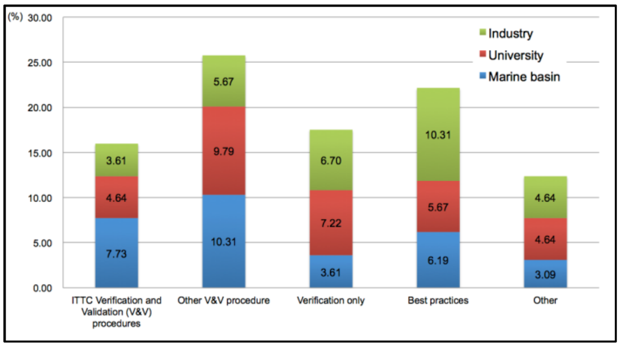



Figure 2.6 Methods used to check the quality of computations

## 2.4 CFD Codes

The questionnaire also asked the characteristic and names of CFD codes used for their applications. The majority of respondents are using commercial codes as shown in Figure 2.7. The figure also indicates that commercial codes are most widely used in industry, while in-house or academic codes are more used in model basins and universities. Figure 2.8 shows the name of codes listed by the respondents.

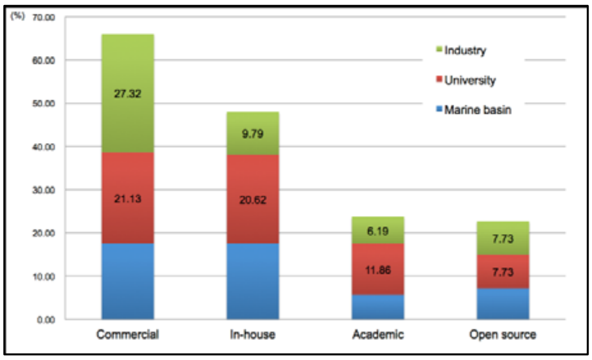



Figure 2.7 Types of CFD Codes

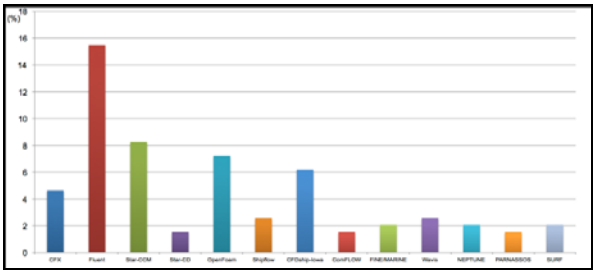


Figure 2.8 CFD codes applied in marine hydrodynamics

### 2.5 Difficulties and Limitations of CFD

The questionnaire asked what are the difficulties and limitations of CFD to achieve wider use and acceptance for application to marine hydrodynamics problems. As shown in Figure 2.9, the accuracy of CFD results is felt as the main problem to use CFD in practice for industry and university respondents, while it is grid generation for model basin respondents. Other areas that are felt as restricting are long turnaround time, especially important to university and marine basin users, and limited confidence in CFD Results.

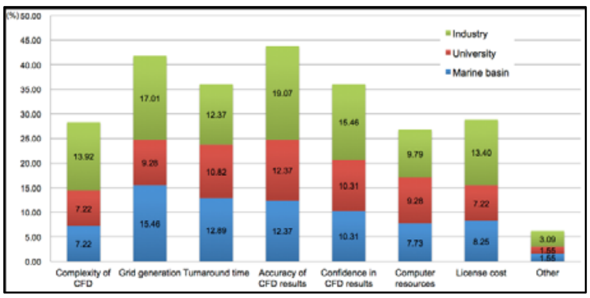



Figure 2.9 Difficulties on application of CFD

### 2.6 Conclusions

The committee prepared the questionnaire to gather information on usability, limitations, and applications of CFD in marine hydrodynamics. The results show that CFD is being used as a tool in the ship hydrodynamics community, and can be used as a baseline to evaluate CFD perception and usage trends in future surveys.

## 3. PHYSICAL MODELLING

### 3.1 Introduction

This chapter reviews the current numerical methods in the area of computational fluid dynamics (CFD) where different numerical techniques for physical processes are described as ‘models’ for the particular process. General descriptions of each model are outlined with suitable terminology introduced as required. The particular models used by CFD that are related to the field of naval architecture are summarised in five parts: Free surface models; Turbulence models; Cavitation models; Propulsion models and Conclusions.

### 3.2 Free surface modelling

The governing equations for the bulk of the flow are the Navier-Stokes equations, and free surface effects appear in the form of boundary conditions. Free surface conditions consist of two conditions: one is the kinematic condition and the other is the dynamic condition.

The kinematic condition constrains fluid particles on a free surface to remain on the interface. In a numerical scheme, this can be implemented in various ways such as interface fitting methods or interface capturing methods.

The dynamic condition forces stresses on air and water to be continuous across the interface. Both normal and tangential components should be considered. Usually in ship hydrodynamic applications, surface tension effects can be neglected, although they must be included in some cases. In case of a single phase flow model, in which only a water region is solved, the normal stress condition yields the pressure on the free surface to be equal to atmospheric pressure and the tangential stresses are zero.

For simple water wave problems, a Navier-Stokes solver with free surface boundary conditions is sufficient to achieve flow field simulations. However, in some cases additional modelling is required. Wave breaking is one of such cases. A bow wave of a blunt ship or waves generated by high speed ships often exhibit wave breaking. Wave breaking is a highly complicated phenomena and it involves free surface deformation and overturning, re-entry, air entrainment and momentum dissipation. For the direct numerical simulation of breaking waves, a multi-phase flow model with flexible interface treatment is required.

For practical applications, simpler wave breaking models can be adopted. One of the models is to add additional pressure on the surface in the breaking region in order to mimic the momentum dissipation (Cointe and Tulin 1994, Rhee and Stern 2002, Muscari and Di Mascio 2004).

In order to simulate flow fields with incident waves, wave models are needed. Wave generation can be achieved by imposing proper boundary conditions on the inlet boundaries. From the numerical point of view, the progression of waves without damping and the non-reflecting boundary condition on the outflow boundary are the issues to be considered. In the physical modelling, the wave generation can be implemented by emulating the wave makers used in actual wave tanks. A second approach is to impose velocity and wave height following theories of ocean waves.

Ambient waves for the reproduction of actual sea environments can be achieved by imposing waves with a given spectrum (Carrica 2008a, Ferrant 2008).

Benchmark data for free surface ship flows collected so far are standard test cases and used in various workshops such as CFD Workshop Tokyo (Hino, 2005) or G2010 Workshop (Larrson, Stern and Visonneau, 2010), which include data for Series 60 (IIHR), KRISO Container Ship (MOERI), DTMB 5415 (DTMB, IIHR, FORCE) Hamburg Test Case (HSVA) and data from the EU projects.

Most of these data sets are for steady flows and wave contours and wave profiles are given together with hydrodynamic forces and moments. Wake distributions at the propeller plane are also provided. Unsteady flow data sets are much scarcer than steady ones. The DTMB 5512 in head seas (IIHR) case is available for wave height and velocity distributions.

Benchmark data for breaking waves is limited. A blunt bow ship data (NMRI) is available for wave contours and hull surface pressure distribution (Hinatsu et al. 2001). There may be some data at full scale with free surface measurements, but publicly disclosed data are limited.

### 3.3 Turbulence modelling

Turbulence modelling has been an important research topic over the last decades. A large number of models have been proposed, tested and applied, but no ‘universal’ model has been developed. Thus one is forced to choose the best model available for each specific application.

The majority of turbulence models are based on the so-called Boussinesq hypothesis, which defines a turbulent or eddy viscosity (as opposed to the molecular viscosity) to account for the effect the turbulence motion has on the mean flow.

Zero-equation, or algebraic models express the eddy viscosity in terms of the mean flow variables and mean flow gradients without solving any additional equations. They are hardly ever used in ship hydrodynamics.

One-equation models solve one additional equation (i.e. in addition to the momentum and mass conservation equations) for the eddy viscosity. Regularly encountered in ship hydrodynamics are models by Menter and by Spalart-Allmaras. These models are sometimes extended with a correction for vortical flow, to improve wake field predictions.

Two-equation models solve two additional equations for the eddy viscosity, one for the turbulence kinetic energy ($k$), and one for its dissipation rate (typically $\varepsilon$ or $\omega$). These models are by far the most popular and have shown to be able to give accurate predictions in ship hydrodynamics, especially certain versions of the $k$-$\omega$ model.

An important class of turbulence models, not based on the Boussinesq hypothesis, are the Reynolds-stress models, and versions thereof. Rather than introducing an eddy-viscosity, they aim to solve the equations for the six Reynolds stress components directly. Apart from that, additional equations have to be solved, since terms in these equations require modelling as well. Consequently Reynolds-stress models are more computationally intensive, and often less easy to converge, compared to the one or two-equation models. However, they contain more physics and can be expected to be more accurate than eddy-viscosity models.

A more recent development is Large Eddy Simulation (LES). Other than the turbulence models discussed above it does not average the Navier-Stokes equations in time, but filters them in space. This results in transient computations on extremely dense grids as they aim to resolve all turbulence motion to a very small scale. Detached Eddy Simulation (DES) is a hybrid method that tries to reduce the required computational effort by solving the (unsteady) RANS equation in the boundary layer and applying LES in the rest of the domain. However, the very high Reynolds numbers encountered in ship hydrodynamics prevents the application of both methods in practical design projects.

### 3.4 Cavitation modelling

Cavitating flow is unsteady by nature and is locally compressible in the vapor-liquid mixture region with extremely high density gradients in transition between vapor and liquid. These features present great challenge for CFD methods. A physically consistent model must take into account the strong interplay between the vapor/liquid two-phase mixture, flow viscosity and turbulence. Existing methods can be classified into three categories: Discrete bubble method; Interface tracking method and Interface capturing method.

Discrete bubble method treats cavitation as an interaction between bubble nuclei and pressure field variations. The change of bubble sizes in response to surrounding pressure is governed by a bubble dynamics equation (e.g. Rayleigh-Plesset equation). The bubbles will grow or diminish as they travel through a decreasing or increasing pressure field. This type of method is mainly applied for cavitation inception, travelling bubble cavitation, and nuclei effects.

Interface tracking method assumes that there is a distinct interface that separates the liquid and vapor region, and the interface is determined by a pressure streamline criterion. The pressure inside the cavity is assumed to be vapor pressure and only the liquid region exterior to the cavity is solved. Methods of this kind are based on potential flow theory and their applications are limited to steady attached sheet cavitation and supercavitation.

Interface capturing method considers the cavitating flow as a mixture of liquid and vapor phases. The phase boundary is determined as part of solution. The most dominant method is called multi-phase Homogeneous Equilibrium Mixture (HEM) approach, where the liquid and vapor phases are mixed homogeneously in the sense that they share the same velocity, pressure and temperature. The mixture medium is treated as one-fluid having a variable density and governed by one set of N-S equations. The cavitation extent is described by a vapor volume fraction. The equation system needs to be closed with additional equation(s) containing a cavitation model to describe the phase transition process, and this equation is usually used to solve for the volume fraction of vapor (or liquid). Three types of cavitation models are available:

(a) Barotropic model that simply relates the mixture density to the static pressure through an isothermal barotropic state law ρ=f(p) (Delannoy and Kueny 1990, Song and He 1998). The barotropic model is unable to predict an important quantity called "baroclinic torque" that is present in the vorticity transport equation and contributes to vorticity production.

(b) Transport equation model that describes the convection of vapor volume fraction, with evaporation and condensation source terms to control the mass transfer between two phases. The source terms are either derived from a simplified Rayleigh-Plesset equation to relate the evaporation with the drop of pressure below the vapour pressure (Singhal et al. 2002, Schnerr and Sauer 2001), or based on some semi-empirical/theoretical formula that are tuned to reflect the cavitation behavior observed in experiments (Merkle et al. 1998, Kunz et al. 2000, and Senocak and Shyy 2002). The HEM approach combined with a transport equation model is the most widely used model today to predict vortex cavitation, attached sheet, and unsteady sheet cavity including the shedding of clouds.

(c) Thermodynamic equilibrium model assumes that the mixture fluid remains in a thermodynamic and mechanical equilibrium. Three equations of state are used to describe the thermodynamic effects for three possible fluid states (liquid, two-phase mixture and vapor) respectively. Together with a set of N-S equations and the energy equation, the system becomes closed (Schmidt et al. 2006). Though the involvement of more number of equations and thermodynamics has limited their application in ship hydrodynamics, these models follow more strictly physical laws and do not use user-defined parameters, thus offering an interesting alternative.

Turbulence handling in a flow solver plays also an important role for cavitation modeling. Incompressible RANS method coupled with a turbulence model has been widely used in the past. However, numerous studies found that the classical turbulence models (e.g. of standard *k*-ε and *k*-ω type) are unable to predict the periodic shedding behavior of transient cavity, unless a modification is introduced for the otherwise over-predicted turbulent viscosity (Reboud et al. 1998, Coutier-Delgosha et al. 2002, Dular et al. 2006 and Li et al. 2008).

An emerging interest is the use of LES due to its rationality to resolve the large-medium scale turbulence structures (Bensow and Bark 2010 and Kim et al. 2008).

### 3.5 Propulsor modelling

Geometrical models define the propulsion system using computational grids which conform to the physical shape of the rotating geometry components. The rotation of the propulsor is defined by a local frame of reference. For open water propellers this is generally the preferred method. Only a single blade needs to be modelled with periodic boundary conditions being used to define the interaction between the blades. A range of different turbulence models and cavitation models can then be used as required.

For fully transient flow calculations the rotational position of the propulsor is incremented at each time step. The flow solution process must redefine the time interpolation process between the two grid components given the relative motion of each. This process provides the most complete description of the interaction between a ship hull and its propeller(s) but the process is generally too computationally expensive for general purpose design. More recently, some new capabilities are being developed which use the Fourier components of the inflow to the propulsor to enable the transient interaction of single blades with non-uniform circumferential inflow conditions to be obtained. These use time-inclining methods to define the periodic conditions for the rotating blade.

For steady state flow calculations a number of different simplifications are used to transfer flow parameters from stationary frames to the rotating frame. These simplifications include: 'frozen rotor', where the propulsor is assumed to be instantaneously frozen in time, 'pitch change' where the inflow to the propulsor is artificially modified in the circumferential direction to match a single blade and 'stage averaging' where the flow parameters are circumferentially averaged surrounding the propulsor. The frozen rotor and stage averaging methods appear to be the most used.

Body force methods define the propulsion system where the influence of the rotating components is applied as additional body forces or momentum source to the underlying computational grid. The computational grid does not conform to the geometry of the rotating components. The body forces or momentum sources can be defined using a large number of different techniques. The forces or sources are defined so that they integrate numerically to the thrust and torque of the propulsor.

One of the most common techniques is to prescribe an analytic or polynomial distribution of the momentum sources. A number of distribution types might be used ranging from a constant distribution to complex functions which define a transient, radially and circumferentially varying distribution of momentum sources. The accuracy of these methods then depends on the complexity of the distribution functions. They also depend on the knowledge of the thrust and torque performance of a particular propulsion system. For some cases the results of geometrical models are used to define the distribution of the momentum sources. This method is often used to obtain the 'powered wake' at self-propulsion by matching the thrust of the propulsor to the drag of the ship hull at a given speed.

Another technique links the distribution of momentum sources to a boundary element method which uses the propulsor geometry to define the distribution of sources. This technique requires the evaluation of the effective wake to couple the two methods. This provides an efficient method for evaluating the performance of a given propeller over a range of operating conditions.

## 3.6 Conclusions

The numerical techniques required by CFD for the physical modelling of viscous ship hydrodynamics have matured sufficiently that predictions of the steady state resistance of a ship hull at model scale can be routinely obtained with reasonable confidence. The numerical techniques associated with free surface and turbulence modelling are sufficiently understood to be used reliably for such applications. Detailed benchmarks have been carried out for representative ship hulls.

Details of the techniques for free surface modelling, such as wave breaking and unsteady RANS, DES and LES based turbulence models require further evaluation before they can be classed as mature. The trends in computing performance are helping the development of these models. Further progress has been made in validation studies of various cavitation models for more realistic and complicated cases. The current trend is to use the transport equation based cavitation models to predict unsteady sheet cavity associated with cloud shedding. It is observed that some modification to the existing turbulence models in RANS solvers seem to be necessary for a successful prediction of cloud shedding. Developments in propulsor modelling are also progressing, with significantly more application of the techniques to representative cases.

It is recommended that further benchmark cases should concentrate on the propulsor and cavitation issues which still need addressing. Benchmark data for unsteady cases and for full scale cases are also required.

## 4. NUMERICAL MODELLING

### 4.1 Introduction

This chapter reviews the current numerical methods in the area of computational fluid dynamics (CFD) where different techniques for solving and discretising the physical 'models' outlined in the previous section. General descriptions of each method are outlined with suitable terminology introduced as required. The particular methods used by CFD that are related to the field of naval architecture and summarized as follows: Solution algorithms; Space and time discretization; Free surface modelling; Grid generation; Solution adaptation; 6DoF and motions; Verification and validation; High performance computing and Conclusions.

### 4.2 Solution algorithms

For ship hydrodynamics applications, the fluid flow of interest – fresh or seawater in low speed regime - is invariably assumed incompressible. Incompressible flows require a special treatment of the continuity (mass conservation) equation, since the fluid density does not appear in the continuity equation. Depending on how continuity is enforced, solution algorithms for incompressible Navier-Stokes equations fall into two categories as follows.

Artificial compressibility method

In this method, the continuity equation is cast into a form akin to one that is widely used for compressible flows. A term involving first-order time-derivative of pressure in combination with a compressibility parameter is added to the continuity equation. Incompressibility is enforced using the concept of "artificial compressibility" (AC hereafter) following the idea originally proposed by Chorin (Chorin, 1967). The resulting equations become a system of hyperbolic equations. Thus, the solution algorithms developed for compressible gas dynamics can be readily applied. The AC method can be considered as a special case of preconditioned compressible flow formulation that the aerospace community has worked on all these years. The system of equations is typically solved in a coupled, implicit manner, which helps convergence and stability of solutions. The AC-based method has been used for a wide range of complex incompressible flow applications (Rosenfeld *et al*., 1991). Among the CFD solvers known to the ship hydrodynamics community based on the AC method are SURF (Hino, 1998) and *Tenasi* (Briley et al., 2006). Although the AC method has been used mostly for steady flows, time-accurate solutions can be obtained using dual time-stepping method.

Projection method

This method uses pressure as a constraint to enforce divergence-free velocity-field. Being more widely used than the AC method, this approach mathematically involves a "projection" of momentum equations (velocity-fields) onto a divergence-free vector-space - hence the name projection method. The projection process yields a Poisson equation as the governing equation for pressure. Since it was first introduced by Harlow and Welch for computing unsteady incompressible flows, several variants of the projection method have been proposed, including the well-known SIMPLE-family (SIMPLE, SIMPLER, SIMPLEC), and PISO method, to name the most popular ones (Harlow and Welch, 1965). In all these methods, the solutions are advanced in time in multiple (fractional) steps. The momentum equations are first solved without pressure or with pressure from the old time-step. Next, the Poisson equation for pressure or, alternatively pressure-correction is solved. Finally, the velocity field is corrected using the new pressure. The solution procedure adopted in the majority of the projection methods is sequential. This segregation or decoupling of the originally coupled equations often make the projection method-based solutions converge more slowly in comparison with the AC-based coupled solvers. Nonetheless, the majority of contributing CFD codes at the Gothenburg 2010 workshop (Larsson et al., 2010) adopted the projection method. This clearly shows the projection method is the main workhorse for ship hydrodynamics applications as far as solution algorithms is concerned.

Other methods

However, other methods exist as well that have successfully been applied in ship-hydrodynamic applications. For instance, MARIN's viscous-flow solver PARNASSOS solves the momentum and continuity equations in their original, fully coupled form. Due to the fully coupled formulation the continuity equation need not be recast in a pressure correction or pressure Poisson equation, but can simply be solved as it is. After discretization and linearization, the three momentum equations and the continuity equation give rise to a matrix equation containing 4*4 blocks, which is solved using preconditioned GMRES. This fully coupled solution has been found to be robust and quite insensitive to the mesh aspect ratio.

### 4.3 Space and time discretization

Spatial and temporal discretization is a central issue when it comes to CFD solvers, since it largely determines not only accuracy but also stability of numerical solutions. The issue of solution stability (or robustness) is especially important for industrial applications like ship hydrodynamics involving complex geometry.

Spatial discretization

Surveying the literature in the area of ship hydrodynamics (Larsson et al., 2010) shows that the CFD codes used by the ship hydrodynamics community predominantly adopt finite-volume (FV) discretization. Furthermore, the majority of modern FV-based codes permit use of arbitrary polyhedral elements, frequently referred to as unstructured grids. Using unstructured grids greatly facilitates computations of applications involving complex geometry. FV discretization can be implemented using either mesh-dual or element itself as control volume. Gradients at control volumes, which are needed for evaluating convective and diffusive fluxes at control-volume boundaries, are computed using either Green-Gauss theorem or least-square method.

In the majority of FV codes, diffusion terms in the governing equations are invariably discretized using what essentially amounts to a second-order central differencing scheme used in classical finite difference (FD) method. For convection terms, a fairly large number of discretization schemes exist today. The more popular ones, especially for RANS computations, are the family of upwind-biased, second-order schemes. The second-order upwind (SOU) schemes widely used today differ from one another in terms of flux-limiter or slope-limiter designed to suppress unphysical oscillations in solutions. Structured mesh-based FV solvers usually offer 3rd-order and $5^{th}$-order upwind schemes. These higher-order schemes provide better accuracy, yet often at the expense of de-stabilizing solutions for industrial applications involving complex geometry, complex flow physics, and poor-quality computational meshes.

Second-order central differencing (CD) scheme for convection discretization is frequently used in large eddy simulation (LES) and direct numerical simulation (DNS) because of its low-dissipation that is critical to accurately resolve small-scale turbulence. However, it is well known that CD scheme triggers numerical instability when cell Reynolds number ($U\Delta x/\nu$) becomes large. Thus, it can cause problems in the form of numerical noise with cases involving fine meshes and small (effective) viscosity, whose use is typical in LES.

The volume-fraction equation, which is employed for interface capturing, requires a special treatment, inasmuch as volume-of-fluid, by definition, behaves like a step-function in the vicinity of free surface. Traditional convection discretization schemes designed for convection-diffusion equations perform poorly in resolving sharp interfaces. Compressive schemes with downwind bias have been found to resolve sharp interfaces much more accurately (Kim et al., 2010).

<u>Temporal discretization</u>

The majority of CFD solvers widely used by the ship hydrodynamics community today use implicit time-marching schemes. Implicit time-marching schemes allow one to use much larger time-step size than explicit time-marching schemes, speeding up numerical solutions for flows with large characteristic time scales. Implicit time-marching, however, requires solutions of system of coupled non-linear equations, which incur computational cost. Explicit time-marching, which forces much smaller time-step size, is rarely used for RANS computations. Its use is rational only for LES and DNS in which one has to resolve time scales of turbulent eddies that require a fairly small time-step size.

In terms of discretization of time-derivatives, first-order backward Euler scheme is often used when steady-state solutions are pursued. Like first-order upwind convection discretization scheme, first-order backward Euler scheme introduces a considerable numerical diffusion into the solutions.

For time-accurate solutions, second-order schemes such as Crank-Nicolson and three-level backward schemes seem to be the most popular choices, as evidenced by the survey conducted at the Gothenburg 2010 workshop. Runge-Kutta schemes with up to $4^{th}$-order have been attempted. However, they are not as widely used as $2^{nd}$-order schemes. This is also commensurate with the observation that second-order spatial discretization schemes are most popular.

### 4.4 Free surface modelling

Numerical modelling of free surface treatment can be categorized into two concepts: one is the Eulerian method and the other is the Lagrangian method.

The Eulerian methods use a function of space and time to define a free surface shape. It includes an <u>interface fitting</u> approach in which a numerical grid is aligned to a deformed free surface shape and an <u>interface capturing</u> approach in which a free surface shape is defined as an iso-surface of a marker function and the grid does not to fit to the free surface.

The Lagrangian approach is also known as particle method. Usually a computational grid is not used at all and the space disretization is made using a large number of particles distributed in a domain. Each particle moves following the local velocity and a free surface shape can be determined from the particle distributions.

Review on the numerical modelling of free surface ship flows is found in Wackers et al. (Wackers et al 2011).

In the interface fitting approach, the wave height function $h=h(x,y,t)$ is used, with $z=h$ being a free surface shape. From the kinematic free surface condition, a partial differential equation is derived and it is solved in the same manner as a bulk flow solution. After the free surface shape is determined, the grid is deformed in such a way that one of the grid planes is aligned to the free surface.

This approach is conventional and many applications can be found in the literature (Hirata et al. 1999, Starke et al. 2010, Alessandrini et al. 1999, for example).The interface fitting method offers a high accuracy because the free surface conditions can be imposed accurately in the exact location of the interface. However, difficulties arise when a free surface shape deforms largely in such cases as very steep waves or breaking waves.

In the interface capturing approach, different marker functions have been proposed: a levelset function, volume-of-fluid (VOF) and a density function.

The levelset function method, first proposed by Osher and Sussman, is defined as a signed distance from the interface. An iso-surface of the levelset function gives the free surface location. The convection equation is used to track the free surface deformation. The level set function is defined in the whole domain, not only on the water region but also on the air region, and its convection requires velocity of the whole domain as well. This is not a problem when multi-phase (air and water) flow is solved in a bulk flow solver. However, if a single-phase flow approach in which only the flow field of the water region is solved is adopted, the velocity in the air region must be extrapolated in an appropriate manner. Several marine applications using levelset methods can be found (Hino et al. 2010, Carrica et al. 2006, Iafrati et al. 2001, Kim et al. 2010, among others).

In the Volume-of-Fluid (VOF) method, the volume fraction of water in each cell is used as a marker function, where the value of unity means a cell is filled with water and zero corresponds to the complete air. An iso-surface of the VOF function of 0.5 defines the free surface in this case. The convection equation for VOF function is solved in the whole domain as in the levelset approach. Usually, a multi-phase flow model is used for the bulk flow in VOF approaches. The density function method is very similar to the VOF approach. The VOF or density function methods is also used in many solvers in marine CFD (Queutey et al. 2007, Manzke et al. 2010, for example)

The interface capturing approaches can be used when the interface deformation is large, although the accuracy of boundary conditions is not as good as the interface fitting method. Another advantage of the interface capturing approach, which is particularly attractive for an unstructured grid method for a complex geometry, is that it does not need re-gridding due to the free surface movement.

In some application areas, such as seakeeping, green water and sloshing, particle methods are becoming more and more popular. In the particle methods, the momentum equations are solved in the Lagrangian manner on the particles distributed in the domain. Each particle moves with its velocity. Several schemes have been proposed for the particle methods, including Smooth Particle Hydrodynamics (SPH) (Oger et al. 2007) and Moving Particle Semi Implicit (MPS) (Shibata et al. 2009). These approaches do not require a computational grid, which is a big advantage over gridded methods. However, the boundary condition on a solid wall needs special care and it is not an easy task to resolve the boundary layer near the wall with particles.

Hybrid approaches are combinations of grid methods and particle methods. A bulk flow is solved in the grid based method and the free surface is tracked by using the particles distributed on a free surface.

In the Eulerian/Particle method by Mutsuda (Baso et al. 2011), the Lagrangian particles are used for free surface model in a Cartesian mesh where a bulk flow is solved. A solid body is modelled by using yet another type of particle based on SPH. The Cubic Interpolation Propagation (CIP) method (Yabe et al. 2001) has been applied to ship flows (Hu et al. 2010). A Cartesian grid system and a VOF-like function is solved by the CIP scheme and the body boundary is treated using virtual particles.

### 4.5 Grid generation

Grid generation is the process by which the fluid region surrounding the ship geometry is sub-divided into a large number of computational cells to obtain the fluid flow parameters. The size, shape and distribution of these computational cells define the resolution of the flow gradients on which the solution and spacial discretization algorithms depend. This influences the accuracy and efficiency of the flow solution algorithms where computational grids with a large number of cells may be more accurate but require significantly more computational resources to obtain a flow solution.

The various methods used to obtain the computational grids use a number of different cell types to provide the basis for the sub-division of the fluid region, typically tetrahedral, prism, hexahedral and polyhedral cells. The methods by which these cell types are distributed within the fluid region are outlined in the following sections.

The simplest form of grid generation is the Cartesian method which simply subdivides the region surrounding the ship hull into regular, rectilinear, hexahedral cells. Although this method is very simple, and is often used as a background grid, it has severe limitations in resolving the flow gradients within the boundary layer surrounding the ship hull. The Cartesian cut cell method uses octree subdivision and inflation layers to resolve the boundary layer using polyhedral cells to provide a more efficient and appropriate process.

More complex forms of grid generation are the structured body fitted methods where hexahedral cells are distorted to fit around a complex shape. This type of method can be defined using single-block or multi-block techniques using curvilinear coordinate interpolation schemes within each block. Multi-block techniques use topological inter-connections to connect the faces of the blocks. Elliptical smoothing algorithms are used to improve the quality of the grids. These methods are in common use for ship hull flow computations as they can readily resolve boundary layer flow around complex geometries.

Unstructured body fitted methods use tetrahedral cells with inflation layers to resolve the flow boundary layer. Octree subdivision, Delaunay point insertion and advancing front techniques are used to create the tetrahedral cells. The tetrahedral cells can be combined to provide more efficient polyhedral cells. These methods can be used for complex geometries but they are less efficient than the equivalent hexahedral methods.

These grid generation methods can be combined using non-conforming grid techniques where specialised schemes within the flow solution algorithm are used to interpolate the flow parameters from one grid to another. This can be done using overset or overlapping techniques where the interpolation is applied across local cell volumes and using interfaces where the interpolation is applied across local cell faces. These interpolation schemes can be applied dynamically to form transient moving and sliding grids to account for the relative motions of the ship hull and the rotation of the propulsion system and appendages.

## 4.6 Solution adaptation

Solution adaptation is the process by which the grid and the fluid flow solution parameters are modified during the solution process itself in order to obtain a more accurate description of the fluid flow. This takes the form of localised refinement processes where *h*-refinement is used to describe processes which modify the grid and *p*-refinement is used to describe processes which modify the solution process. The purpose of the refinement process is to reduce the numerical errors associated with the discretization of the flow parameters. All of these localised refinement processes require the evaluation of adaptation markers which are used to define the regions in space (and time for transient cases) where the adaptation processes are taking place.

Grid refinement processes (*h*-refinement) use the adaptation markers to refine the grid so that the local spacial resolution is increased. This can be achieved by grid point insertion where the number of computation cells is increased or by grid point movement where the number of cells remains constant and the vertices of the cells are moved. Additional computational cells can be created by sub-dividing existing grid cells or by regenerating the local grid region. These sub-division processes may be isotropic (where the aspect ratio of the cells do not change) or anisotropic. Solution refinement processes (*p*-refinement) locally modify the order of accuracy of the solution scheme, so that a 3rd, 4th or 5th order method is used instead of a 1st or 2nd order method. Typically the *h*-refinement methods are more readily used with unstructured codes and the *p*-refinement methods are more readily used with structured codes, although there are exceptions, especially for finite element (FE) based methods.

The evaluation of the adaptation markers is still the subject of considerable research and development. The simplest techniques use geometric descriptions to define the regions to refine the grid or solution, for example wake planes or wave surfaces based on the experience and knowledge of the user. Some techniques use solution gradients to define the adaptation markers so that regions of high gradients in the velocity, pressure and turbulence are refined. Other techniques use solution markers where additional parameters are convected with the flow to track. Finally, error estimators can be used to define the adaptation markers. These error estimators evaluate higher order terms of the discretization schemes to obtain numerical estimates of the errors associated with each of the terms in the solution equations. Such techniques are complex and require considerable detailed knowledge of the numerical errors. The geometric methods can be used before a flow solution is obtained to provide an initial guess at appropriate regions of refinement but in most cases the adaptation process is carried out after an initial flow solution has been carried out. An iterative process is then used to successively refine the grid and flow solution.

For hydrodynamic applications care is required to ensure that adaptation processes provide valid grids due to the high aspect ratio computational cells on curved surfaces that are required to resolve the hull boundary layer at high Reynolds numbers.

## 4.7 6DoF and motions

Predicted motions are used to compute sinkage and trim in calm water resistance tests, pitch and heave in waves, trajectories, and general 6DoF motions under varied seakeeping and manoeuvring conditions. One of the first examples of ship computations with motions can be found in the 90s (Sato et al. 1999). The motions are imposed or computed solving the rigid body equations for the ship, either in Euler angles or quaternion formulations (Fossen 1994).

To solve the rigid body equations it is necessary to obtain the instantaneous forces and moments acting on the object. This is generally done integrating the contribution of pressure and viscous forces on each cell on the solid body. This approach is accurate but it may be complicated for immersed boundary and overset methodologies. A second approach is to balance linear and angular momentum on the fluid, including unsteady terms. This approach is easier to implement but it is less accurate since any inaccuracies in the momentum balances will be attributed to forces and moments on the object.

A common approach to account for motions is to solve the fluid flow equations in the ship system of reference and add body forces to account for the non-inertial accelerations. The biggest advantage is that the grids do not need to be deformed or moved during the computation, but important features (such as the free surface) may shift to poor quality grid regions (Sato et al. 1999 and Cura Hochbaum and Vogt 2002, among others).

A second approach to simulate motions is to compute the fluid flow in an inertial coordinate system and move the grids following the object. Advantages are that it is easier to maintain dense grids where needed, and that multiple ships can be simulated. To move the objects, deformable (Ohmori 1998) and overset grids (Carrica et al. 2007) have been tried, in addition to grid regeneration, grid deformation and sliding grids typically available in commercial codes. These methods to deform the grids can be used in conjunction with the body force approach to maintain grid quality as the motion progresses.

Though the techniques are available, most computations in the literature deal with a reduced set of motions 1 DoF (typically roll decay), 2DoF (sinkage and trim, pitch and heave in waves), 3DoF (manoeuvring trajectories constrained from pitch, heave and roll, PMM predicting pitch, heave and roll), etc.

In the Gothenburg 2010 CFD Workshop (G2010) several cases involving motions were included; while in the previous workshop in Tokyo (Hino 2005) motions were absent mostly because of the lack of capability in the CFD tools at the time. In particular, cases involving prediction of sinkage and trim in calm water, pitch and heave in head regular waves and roll decay (initial roll angle 10 degrees) in calm water were part of G2010 for diverse geometries (tanker, containership and surface combatant). Results from G2010 show that different CFD methodologies are perfectly capable of predicting the ship motions for the aforementioned cases. For all these cases the resulting motions are of small or moderate amplitude.

Also in G2010 two cases of self-propulsion of the KCS containership were computed using discretized rotating propellers by the participants. Sliding grids and overset methodologies were used, and the results of the self-propulsion point and associated factors were excellent.

Computation of large-amplitude motions requires more elaborated techniques. Though unstructured re-gridding is sometimes used, dynamic overset grid technology appears to be the trend to solve problems like broaching, parametric roll, ship-ship interaction and others in which motions are so dramatic that moving refinement grids are necessary to follow the objects. This trend is also currently observed in the aerospace industry. These techniques for large-amplitude motions enable computations with moving rudders and propellers in a seaway at full scale, as well as simulations of moving ships launching or retrieving service vehicles, deploying platforms, refuelling, etc.

### 4.8 Verification and validation (V&V)

The basic assumption for Verification and Validation (V&V) methods is to have a set of CFD solutions that are in or enough close to the asymptotic range. The idea is then to use methods based on Richardson extrapolation to estimate quantitative numerical error/uncertainty for grid and time-step convergence. The grid convergence index (GCI), widely used (e.g. ASME), can be used to estimate the uncertainties. Richardson extrapolation methods are difficult to perform because (i) all the solutions must be close to the asymptotic range (otherwise the estimated order of accuracy $p_{RE}$ approaches the theoretical order $p_{th}$ with oscillations) and (ii) require three or more refined high-quality grids (often too expensive for industrial applications). The non-smooth grid convergence problem may be resolved using the least-squares method, which requires solutions for more than three grids and of course is even more expensive. In a relatively recent study Eça and Hoekstra presented a least-squares version of the GCI method (Eça and Hoekstra 2006).

The correction factor (CF) method (Stern et al., 2001) uses a variable factor of safety (FS). The method is validated (using analytical benchmarks) for a CF less than 1, whereas for factors larger than 1 it is obtained by assuming that it is symmetric with respect to the asymptotic range. GCI and CF methods have some issues too: uncertainty estimates for $p_{RE}>p_{th}$ are too small in comparison to the corresponding values for $p_{th}>p_{RE}$.

Xing and Stern recently developed a FS method for solution verification (Xing and Stern 2010), with different error estimates, a better distance metric to the asymptotic range and removing the assumption that the factor of safety is symmetric with respect to the asymptotic range. They perform a statistical analysis of 25 computational samples taken from the literature, covering fluids, thermal, and structure disciplines.

As a more general comment, this Committee foresees some difficulties in a near future in updating intact the ITTC procedure 7.5-03-01-01. The problem is not a general consensus on the existing procedure or on other proposal, but the general approach of using grid studies. It is indeed not obvious what to do in complex cases, such as unstructured, overlapping or adaptive grids. It is a clear trend that the more robust and reliable the codes evolve, the more the users want to solve complex cases (e.g. torpedo launch from a submarine), moving toward more complex grid types (e.g. overlapping, sliding) for which the procedure 7.5-03-01-01 cannot be extended.

Finally we want to stress the necessity of updating the experimental databases used for CFD validation. New optical measurements of already tested flow configurations show new details and pose new challenges to the codes. An example was recently given at Gothenburg CFD Workshop 2010: the well known Pitot flow data past the DTMB 5415 (Fr=0.28 straight ahead, calm water) was challenged with up-to-date fine grid URANS and DES simulations that found a new vortex in the bulb region which wasn't present in the data taken more than 10 years ago with a 5-hole Pitot tube.

### 4.9 High performance computing

As computers increase processor speed by a factor of 3 every 4 years, overall parallel performance increases by 10 every 4 years. This trend of massively more cores with little improvement in per-core performance is expected to accelerate, marking the importance of parallel capability for CFD.

High-Performance Computing (HPC) efforts are geared towards two different goals: run production jobs faster, and improve the capability of running larger jobs that can resolve more physics with less reliance on modelling. These two are termed strong and weak scalability, respectively.

Scalability studies of free surface CFD codes are scarce and dependent on hardware, and thus conclusions are difficult to reach. Incompressible flow codes tend to roll-off in strong scalability tests when large numbers of processors are used (Kremenetsky 2008, O'Shea et al. 2008, Bhushan et al. 2010), while compressible codes tend to scale better since the solution of a pressure Poisson equation is not needed (Gicquel et al. 2008). The speed up is typically 60 to 80 % of the linear speedup for ideal strong scalability for 1000 processors. Weak scalability is usually more easily achieved and has been the focus of most developments.

Static ship computations of hundreds of millions of grid points have been reported for curvilinear and Cartesian grid solvers (O'Shea et al. 2008, Bhushan et al. 2010), while dynamic moving computations up to 70 million grid points were performed (Carrica et al. 2010a). These computations enable a degree of detail in the flow physics that cannot be achieved with coarser grids, allowing the use of more accurate turbulence models like DES and LES. Moving computations are harder and limited by the need of re-gridding or computation of overset connectivity; see section on 6DoF and motions. Computations with motions are routine for grids ranging between 5 and 25 million grid points.

New promising numerical techniques and hardware technologies are rapidly changing the landscape of high-performance computing. Super-scalable Cartesian grid solvers are breaking the 1 billion grid point limit with distributed memory platforms (Dommermuth 2010, Wang 2010), and soon 10 and 100 billion grid points will be possible.

Graphic Processor Units (GPUs) suggest that large-scale CFD computations will soon be available for massive numbers of users with limited resources. GPUs capable of great computational power in a desktop size are available, and software is being developed that will enable harnessing of that power at a fraction of the present cost per CPU hour.

### 4.10 Conclusions

Over the past two decades, high-performance computing using massively parallel machines and advances in numerical methods have dramatically changed the landscape of computational ship hydrodynamics, impacting ship design in significant ways. Spatial resolution of typical CFD solutions nowadays made possible by the tremendous computing power has greatly reduced numerical error and uncertainty. The fidelity of CFD is fast approaching that of model tests. Furthermore, solution turnaround times for RANS computations have also been dramatically reduced. Advances in gridding techniques such as hybrid unstructured grids, overset grids, and adaptive mesh refinement have empowered CFD practitioners so that they can now properly tackle ship hydrodynamic applications involving complex geometry such as fully-appended ships. We see an increasing number of transient simulations being conducted in attempts to resolve unsteady flow-fields and/or to predict ship motions using 6-DOF motion solvers. With proliferation of CFD software, whether in-house or commercial ones, quality assurance via verification and validation (V & V) has become an important issue.

Despite the progress, we still have significant challenges ahead that have to be addressed before CFD can impact a broader range of practical ship applications. One immediate challenge comes from more compute-intensive application areas like seakeeping that requires an extremely long solution (simulation) time and a very large parameter space (operating conditions) to be covered in simulations. For those applications, the speed of present day CFD solutions is considered far too slow to satisfy the requirement in terms of solution time and to impact design at an early stage. Thus, the challenge lies with how one can speed up CFD solutions by orders of magnitude. Although one should continue innovating numerical methods, easier solutions are likely to be found in taking advantage of faster computers, for instance, using next generation of massively parallel, multi-core machines.

## 5. TRENDSIN CFD FOR NAVAL ARCHITECHTURE APPLICATIONS

### 5.1 Introduction

This chapter summarizes ongoing research efforts toward the development of efficient numerical tools in the area of computational hydrodynamic analysis and design of ships, reporting trends in research and experience in industrial applications as emerged from the literature of recent years. The section outlines the trends that have been observed in each of the traditional Naval Architecture applications: Resistance; Propulsors; Propulsion; Manoeuvring; Seakeeping; Ocean Engineering; Others and Shape Optimization.

### 5.2 Resistance

This section reviews applications of CFD in a category broader than resistance that includes predictions of not only resistance (drag) on ship hulls but also other aspects intimately related to resistance prediction including local flow-fields (e.g., boundary layer and wake), wave patterns, and sinkage and trim.

Resistance prediction

Prediction of resistance (drag) of a ship is the most mature – oldest - application of CFD in ship hydrodynamics. Fidelity of resistance prediction has been significantly improved over the years, although it varies depending on types of ships and operating conditions.

The results of Gothenburg 2010 Workshop on CFD in Ship Hydrodynamics (Larsson et al., 2010) provide the state of the art in resistance prediction. A total of 89 predictions were submitted of resistance for three ships including KVLCC2, DTMB 5415, and KCS at several Froude numbers, in fixed and free conditions, and with and without an operating propeller. The number of submissions is by far the largest in the series of the workshop, providing an invaluable statistical database that helps us assess the state of the art in predicting resistance of ships.

Surveying the workshop results reveals that resistance of the model-scale ships selected for test cases can be predicted, on average, within a few percents from measurements made in towing tanks. 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 ascribed 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 maneuver, 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.

There are a few technical enablers behind this progress in resistance prediction. Unquestionably, the first and foremost is the today's high-performance computing (HPC) backed up by ever-increasing computing power and parallelism in computing. Thanks to the tremendous computing power, RANSE computations on large grids with a few to tens of million elements are commonplace these days. Using much finer grids than before, at least by an order of magnitude than a decade ago, has dramatically reduced numerical error in CFD solutions, significantly improving spatial resolution of fine details of turbulent boundary layer and wake around ships, and resistance prediction. It is useful to note that high-order discretization schemes have not played a significant role in reducing numerical error in this regard. Almost all of CFD practitioners in the ship hydrodynamics community, including the participants of the 2010 workshops, seem to have stayed with second-order schemes. Thus, in computational ship hydrodynamics, *h*-refinement rather than *p*-refinement has been the main driver behind improvement of spatial accuracy.

Another enabler is the advances in gridding technique. First, one clearly noticeable trend in gridding nowadays is wide-spread use of unstructured grids. The ship hydrodynamics community has embraced unstructured meshes mainly motivated by ease of gridding for complex geometry frequently encountered in real ships - fully appended ships. Unstructured meshes are often associated with triangular (2D) and tetrahedral (3D) meshes. Yet the majority of modern FV-based CFD codes supporting unstructured meshes can take arbitrary polyhedral elements, including tetrahedra, hexahedra, prisms, wedges, pyramids, and elements with arbitrary number of faces. Among the amenities offered by unstructured grid-based FV methods is adaptive mesh refinement (AMR). Noteworthy in this regard is that several participants at the 2010 workshop used AMR strategy, albeit simple, in which blocks of finer meshes are embedded around ship hull and free surface. Despite a modest number of elements, the qualities of their predictions were among the better ones. In the same vein, overset grids seem to enjoy popularity among some practitioners. Overset grids are useful for moving-body and multi-body applications such as computations in dynamic (free) sinkage and trim conditions. Furthermore, overset grids provide a convenient means of locally refining grids by being able to embed blocks of finer grids.

Lastly, the impact of turbulence modeling on resistance prediction is worthy of a few words. The issue of turbulence modeling remains still relevant in ship hydrodynamics. In the past two decades, the majority of CFD practitioners in the ship hydrodynamics community seem to have settled with two-equation-based, isotropic eddy-viscosity models (EVM) for RANSE-based computations. The family of $k$-$\omega$ models appears to be the more popular ones. Several participants at the 2010 workshop reportedly used explicit algebraic Reynolds stress models (EARSM). Numerical evidence from various sources including the 2010 workshop (Larsson et al., 2010) shows that resistance prediction is not as much affected by the choice of turbulence models as predictions of characteristic features of local flow-fields. Interestingly enough, a few participants, who are commercial CFD users, used Reynolds-stress transport models (RSTM).Last few years have also seen publications based on detached eddy simulation (DES) and large eddy simulation (LES). However, efficacy of these high-level turbulence simulation techniques has not been convincingly demonstrated yet to the ship hydrodynamics community.

An important issue related to turbulence modeling is treatment of wall boundary conditions. Thanks to today's computing power, it has become feasible to resolve near-wall region all the way to viscous sublayer. Nonetheless, using wall functions can be a viable alternative, in as much as flows around ship hulls are largely attached, equilibrium turbulent boundary layers. Relatively coarse near-wall mesh typically adopted for wall function approach greatly reduces numerical stiffness, giving greater stability and faster convergence of numerical solutions. The fidelity of predictions of resistance and flow-fields using wall function approach has been demonstrated by Kim et al. from NSWCCD at the 2010 workshop.

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 the difficulty of finding systematic and well-documented studies in the open literature. The situation is similar for fully-appended ships with rudders, bilge keels, shafts, and struts for the same reason. We surmise that prediction error for unconventional ships or real ships with all design details would be somewhat larger than with conventional ships or bare hulls mainly due to potentially more complex flow physics, difficulty of generating high-quality mesh, and overall lack of experience in computing such flows. The CFD capability for some of these “more difficult” hulls can be gleaned from the literature (Wackers et al. 2010). With all the enabling technologies discussed earlier, the fidelity of CFD predictions for them will improve as they receive more attention from the community and its experience accumulates.

In recent years, full-scale resistance prediction, the ultimate goal of CFD, has received a renewed interest from the community. Despite the community’s near-obsession with scaled models (for a good reason – we tank-test scaled models!), we see an increasing number of attempts to compute full-scale resistance and associate flow features such as nominal wake-fraction. Understandably, limited full-scale data is the major difficulty in assessing prediction accuracy. Wall function approach would be a practical one due to difficulty of resolving viscous-sublayer at full-scale Reynolds number ($10^8$ – $10^9$). However, thanks to today’s tremendous computing power, we see more and more attempts to resolve viscous sublayer. Having both model-scale and full-scale resistance predictions enables one to study scale-effects of resistance components and wake. Particularly being significant to the ITTC, quantification of scale effects in resistance using high-quality CFD computations will help us to evaluate and potentially improve the ITTC-recommended procedure for extrapolating model-scale resistance measured in towing tanks to full-scale (Schweighofer et al., 2005).

One important issue yet to be addressed while working toward full-scale resistance prediction is that of surface roughness. Computations for both model-scale and full-scale ships including the effects of surface roughness have been carried out (Eça et al. 2010).

Wave Pattern

For surface ships, accuracy with which wave pattern around hulls can be predicted is of great concern, in as much as accurate prediction of waves around ship hull is often regarded as a precursor to accurate prediction of wave resistance. The community has largely adopted interface-capturing approach based on volume-of-fluid (VOF) and level-set (LS) approaches. These two competing methods are able to capture breaking waves that are important for high-speed surface ships.

At the 2010 workshop, the contributors were found to split almost equally between VOF and LS approaches. In terms of accuracy of predictions, VOF and LS approaches appear to be on par with each other, although there is ample evidence in the literature that VOF predictions are more sensitive to discretization of advection term of VOF equation. It is known that using downwind-biased advection schemes is beneficial to resolving sharp interface more. The major factor determining the accuracy of predicted wave elevation seems to be mesh resolution, which was the main conclusion drawn by the workshop organizer based on a careful survey of the meshes used by the participants.

In terms of turbulence modelling, two-equation based RANSE seems to be sufficient a wave Kim et al.’s (NSWCCD) contribution showed that turbulence modelling affects predicted wave patterns very little. The overall improvement of the predictions submitted for the workshop over the last two workshops is thus attributable to finer mesh resolutions adopted at the 2010 workshop.

In summary, modern CFD techniques can predict overall wave patterns in near fields quite accurately as long as proper mesh resolutions are used. Physically more complex wave phenomena such as wave breaking pose challenges

Local Flow-Fields

Importance of being able to predict local flow-fields has been well recognized among naval architects. Main concerns are with turbulent boundary layer, vortices emanating from hull boundary layer, and wake. As in the case of resistance and wave pattern prediction, much progress has been made in the last two decades towards more consistent and accurate predictions of local flow-fields. The main driver behind this progress is again greatly reduced numerical error made possible by use of sufficiently fine meshes. As opposed to resistance and wave pattern predictions, turbulence modeling also plays an important role in predicting local flow-fields.

The 2010 Gothenburg workshop results again provide a useful snap-shop of the state of the art in predicting local flow-fields. The participants were asked to submit local mean velocity fields and turbulence quantities for KVLCC2, DTMB 5415, and KCS. The overall agreement between computations and experiments in terms of mean velocity distributions is fairly good. Compared to the results presented at the 2005 workshop, the level of agreement between computations and experiments has been significantly improved. For all these three cases, the characteristic features of the mean velocity fields associated with bilge vortices were captured by the majority of contributors. On the average, a few million elements were used, which explains the improved predictions. The majority of contribution based on isotropic eddy-viscosity models (EVM) which were found to under-predict the bulge and roll-up in the contours of axial velocity contours. Explicit algebraic Reynolds-stress model (EARSM) seems to closely reproduce the characteristic features of axial mean velocity in the hull boundary layer and at the propeller plane. In addition, EARSM also captures normal stress anisotropy that is significantly large in the propeller plane, more specifically in the core of the bilge vortex as revealed by the experiments,

It was the first time at the 2010 workshop that there were submissions based on LES or DES approaches. By all their appearances, however, LES and DES do not seem to be ready yet for practical ship hydrodynamic applications, especially the cases selected for the workshop – largely attached turbulent boundary layer at high Reynolds number. Even with a huge number of computational elements – up to 300 million, the quality of local flow predictions by LES and DES was embarrassingly bad. Due to very fine meshes used, the LES and DES results tend to resolve fine scales of the flows. However, they were grossly exaggerated because of the under-resolved turbulence as discussed by the workshop organizers

One thing noteworthy from the 2010 workshop is that unstructured grids can provide the same level of accuracy (FreSCo, SURF, and NavyFOAM).

Other issues

Drag reduction using various means including polymer, micro-bubbles, and air layer has been pursued for some years. Recently, the Office of Naval Research (ONR) in U.S. has supported experimental and numerical researches on drag reduction using air layer. Kim and Moin (2010) demonstrated a two-phase modelling capability to predict air layer drag reduction on a flat plate.

### 5.3 Propulsors

This section reviews of the application of CFD to the different propulsion systems which are currently used in ship hydrodynamics.

Open water propellers

The application of CFD to open water propellers can be regarded as developed into a reasonably mature capability. The application of CFD to various propellers operating in open water conditions has been demonstrated by a number of authors (Streckwall 2008, Watson 2008, Gaggero 2009). The number of these applications is illustrated by the 1st and 2nd Symposium on Marine Propulsion (SMP 2009, SMP 2011) in which many papers outline the issues associated with grid generation, turbulence modelling and cavitation modelling required to reliably predict thrust, torque and cavitation performance. For example, transition modelling and propeller skew are shown to influence scale effects (Muller 2009, Krasilnikov 2009). Cavitation modelling continues to be an area of ongoing application, in particular the VIRTUE projects (Streckwell 2008, Salvatore 2009) which compare results from the various models and methods using a common test case.

Operating propellers behind ship hulls and with shafts/brackets

Applications of CFD to operating propellers in the presence of the ship hull wake are increasing rapidly as the techniques required to perform the calculations have been developed and the computational resource increases. Results from the Gothenberg 2010 workshop (Larsson et al, 2010a and b) for the KCS self propulsion test case shows that a full range of these methods are being applied. These methods use the evaluation of the ship wake and the loading of the propeller to provide integrated evaluation of the powered wake of the ship hull. Applications with shaft and brackets (Muscari et al 2009, Carrica et al. 2010b for example) are using overlapping grid techniques which are readily used to define the complex geometry and flow interactions. Similar techniques are also used to provide the interaction between submarine hull forms and the propeller (Alin et el, 2010, Liefvendahl, 2010). Further details are given in the following section on propulsion.

Waterjets

Application of CFD to waterjets is demonstrated by the development of the AX-WJ1 and 2 test cases (Lindauu el 2009, 2011, Schroeder et al, 2009, Kim and Schoeder 2010) that calculate the development of cavitating flow around a stator rotor water jets for a single advance ratio for a range of cavitation numbers. Other similar applications include the optimization of a linear waterjet (Steden et al 2009) where the design parameters for the shape of the propeller, duct and stator are all controlled using an optimization technique which uses a mixture of potential flow methods and RANSE based CFD to optimize the efficiency.

Podded propulsors

Examples of the application of CFD to podded propulsion systems have been used to illustrate the influence of fillets around the strut (Oszu, 2010) where several fillet designs were investigated using RANS methods. Different strut designs for an azimuth thruster unit were evaluated and compared with measured efficiency values (Funeno 2009) giving total performance assessment of the whole units. Details of the hydrodynamic interaction between the nozzles and gear housings were examined by parametrically varying the diameter of the gear housing. The open water performance of a podded propulsion unit for a range of advance ratios (Xingrong 2009) has been predicted using unsteady techniques showing good correlation with measurements.

Ducted propulsion

There are several examples of the application of CFD to ducted propulsion systems. These include development of new nozzle designs (Minchev 2009), development of rimmed propellers for small craft (Chapple 2009) and the Reynolds number scaling of nozzles and ducts (Zorn 2010, Bulten 2011) which also shown comparison of the wake fields generated at model and full scale. In addition to primary propulsion systems, CFD is being applied to secondary propulsion systems, such a bow thrusters.

Interaction effects between propulsors and appendages

There is currently a trend towards the use of CFD to investigate the use of wake adaptation devices, such as vortex generators, upstream ducts and stators to improve propulsor efficiency and reduce pressure pulses (, Simonsen 2009, Dymarski 2011, Hafferman 2011, Heinke 2011, Hollenbach 2011, Mewis 2011 and Zondervan 2011)

A number of propulsion and appendage configurations are considered for a double ended ferry configuration (Minchev 2011) where CFD is used to align propulsion units with thruster head boxes to the flow as well as optimization of the hull form. Interactions between propulsion systems and the rudder are also investigated (Carlton 2009, Simonsen 2010).

### 5.4 Propulsion

In order to perform self-propulsion computations or to study hull-propeller-rudder interaction a CFD method used in double-model or free surface resistance computations is usually taken as a basis. The extra requirements for the method are to model the propellers or propulsors and control surfaces. Coupling a RANS viscous solver and a potential flow-based method for prediction of propeller flow effects is widely used. The incoming flow velocity on the propeller (actuator disk) plane obtained from the viscous-flow solver becomes the input data for the potential-flow solver. The propeller effects are taken into account in viscous flow solver as body forces. The converged solution is achieved in iterative manner or the propeller during RANS solver time steps. Depending on the method the axial components and possibly also tangential and radial components of propeller induced velocities are used. The potential flow propeller codes vary from lifting line to panel methods. In some potential flow methods also the viscous flow along the propellers blades is also taken into account. Also full viscous RANS methods are used to calculate the propeller effects.

Fully discretized CFD computations have been performed for self-propulsion and hull-propeller-rudder interaction studies. When a full RANS solver is used a sliding interface is needed to connect the rotating frame to the fixed frame of the hull part. Another technique is to use moving overlapping grids. The RANS code has to be also capable to compute time accurate solutions.

In the Gothenberg CFD Workshop for ship resistance and propulsion (Larsson et al, 2010a and b) there were two computational cases for self propulsion. In the first case computations were requested for the KCS hull at fixed hull condition and for $Fr$=0.26. The experimental data for validation was available for a 7.3 m model without rudder (case 2.3a, data from MOERI). The self propulsion prediction was requested for full scale and the skin friction difference due different Reynolds numbers was kept the same as in the model tests. In the second case the model was 4.4 m long but it was allowed to sink and trim, and a rudder was fitted (case 2.3b, data from FORCE). In the latter case the computations was requested at model self propulsion point without skin friction correction.

There were two ways to simulate the flow field at the self-propulsion point. Most popular was to load balancing by varying propeller rpm, while the other alternative was to use directly the rpm from the model tests and compute the towing force.

17 different computations were conducted for the first case. Both actual propeller (9 computations) and body force approaches (8) were used (see Table 1).

Table 5.1. Summary of self-propulsion calculations (Larsson et al. 2010). A: actual propellers; BP: Prescribed body force; BL: Lifting line; BS: Lifting surface; BX: Other body force)

When comparing the contributed computations for *KT*, *KQ* and *n*, the relationship between grid size and accuracy shows that the maximum scatter in grids larger than 10 Mcells is around +- 7%, 5% and 2%, respectively, while for grids smaller than 10 Mcells it is within 19%, 18% and 6%. For the towing force estimate using model test *n* there are very few entries and the largest error is for an 11.5M grid.

When the actual propeller results were compared with those from modeled propellers, a clear trend of smaller scatter for the actual propellers was found in $K_T$, $K_Q$ and $n$ values. The difference is particularly large for $K_Q$. For the mean error there was no clear trend. The actual propeller exhibits a considerably smaller error in $K_Q$, but for $K_T$ and $n$ the errors were slightly larger.

The mean values of all computations, may give a general indication of the accuracy obtainable in self propulsion predictions. For KT the mean error is 0.6%D and the mean standard deviation 7%D and the corresponding values for KQ are -2.6%D and 6%D, respectively. The predicted *n* for a given SFC has a mean error of 0.4%D and a standard deviation of 3.1%D, while the values are larger for the towing force for given n: -7.8%D and 8.7%D, respectively.

| Case no. | Classification | Group | Prop. model | E%D | | | |
|---|---|---|---|---|---|---|---|
| | | | | *KT* | *KQ* | *n* | *RT(SP)-T* |
| 2.3a | Given n, actual propeller | CSSRC | A | 0,06 | -1,39 | - | -8,03 |
| | | MARIC | A | 4,12 | -2,88 | - | -4,12 |
| | | SNUTT | A | -1,94 | -7,99 | - | -3,43 |
| | | SSRC(1) | A | 3,35 | -0,35 | - | -8,89 |
| | | TUHH-FDS&ANSYS | A | 6,47 | -0,42 | - | -14,38 |
| | Given SFC, actual propeller | CTO | A | 11,65 | 1,77 | -3,16 | - |
| | | IIHR | A | 0,65 | -2,81 | -1,27 | - |
| | | SSRC(2) | A | -1,59 | -3,82 | -2,11 | - |
| | Given SFC, modeled propeller | IIHR/SJTU | BP | 2,4 | 1 | 0,7 | - |
| | | MARIN | Body f. | -4,7 | -7,3 | 2,6 | - |
| | | MOERI | BS | 1,76 | 0,66 | -1,11 | - |
| | | NMRI | BX | -6,53 | -16,32 | 5,68 | - |
| | | South/QinetiQ | BP | -18,92 | -17,99 | 1,49 | - |
| | | SSPA | BL | -5,34 | -6,26 | 2,34 | - |
| 2.3b | Given SFC, act. or mod. propeller | IIHR | A | 6,7 | 5,1 | -2 | - |
| | | MOERI | BS | 12,13 | 12,55 | -3,87 | - |
| | | SSPA | BL | -0,18 | 2,45 | 4,98 | - |

Table 5.2. Error statistics, Cases 2-3a; the fixed 7.7 m model case and 2.3b; the free 4.4 m model case.

Zhang studied viscous the flow around the container ship KCS with operating propeller (Zhang 2010). The rotating propeller was simulated using both body force approach and real rotating propeller using sliding mesh technique. In the comparison at one self-propulsion point the actual rotating propeller method predicted the axial velocity field better and in more detail than the body force approach. For cross flow both methods and experiments gave qualitatively similar results. The $K_T$ and $K_Q$ values at self propulsion point were well predicted when compared to the measurements.

Choi et al. studied resistance and propulsion characteristics of several commercial built ships. (Choi et al. 2010 and 2009). The viscous-flow solver used was Fluent 6.3 and the potential-flow solver was based on the vortex lattice method for unsteady flow analysis around a propeller. An asymmetric body force distribution was utilized for the actuating propeller. The exact self propulsion point in full scale for model scale computations was obtained using load variation and interpolation. The skin friction correction was based on the ITTC-78 performance prediction method but it includes a CFD-EFD correlation coefficient. The propulsive computations were made for double-model hull and the ITTC-78 method was used for full scale prediction. The computational predictions brought out similar tendencies with experimental predictions. They concluded that the most problematic elements to predict precisely are trimming, propeller geometry and free surface. The differences between CFD and EFD predictions were -3.2 % to 2.3 % for $n_S$ and -9.0 % to 0.2 % for $P_{DS}$.

Xing et al. introduced a procedure to perform resistance and propulsion computations for a wide range of velocities in a single run (Xing et al. 2008). In the procedure the model speed was accelerated using small steps and within each step a quasi-steady solution was obtained. The computations were performed for the fully appended Athena hull using the CFDShip-Iowa-v4 solver. A simplified axi-symmetric body-force propeller model with axial and tangential components was used for propeller simulations. The predicted $Fr$ using CFD compared to $Fr$ of EFD results for same RPS was within 2.1 %. The sinkage and trim differences were less than 11 %.

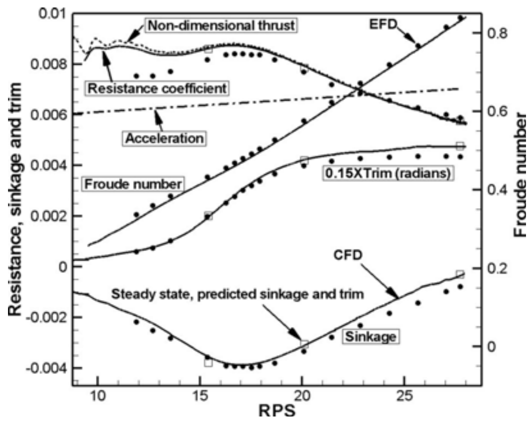



Fig 5.1. Whole powering curve for a slow acceleration of a fully appended Athena hull as a function of RPS. Solid circles EFD data; open squares steady state computations; lines predicted quantities (Xing et al 2008).

Carrica et al introduced a method for self-propulsion computations where the moving parts are gridded using dynamic overset grids (Carrica et al. 2010b). The RPS of the propeller was controlled by a speed controller which finds the self propulsion point when the targeted Froude number is reached. CFDSHIP-IOWA-V.4 solver was used in the computations. The method was applied for three ship geometries: the KVLCC1 tanker, the DTMB 5415 surface combatant and the KCS container ship. The computations required significant resources using 50-160 processors for about a month. The propulsion coefficients and parameters showed good agreement compared to experiments. The largest error was less that 3.7 %.

Han's objective in his PhD thesis was to numerically simulate, analyse and automatically optimize the interaction between a ship hull, a propeller and a rudder (Han, 2008). He used the RANS method in the code SHIPFLOW, the effect of rotating propeller was simulated via body forces by a lifting surface method or a lifting line method when there was a rudder behind the propeller. The hull grid had 2.05 million cells and the overlapping grid around rudder 0.76 million cells.

In hull/propeller/rudder interaction studies model tests of a chemical tanker were used for validation. The effect of rudder distance from the propeller was also investigated. The wake fractions predictions were very close to the experimental values. The computed absolute values of other propulsive coefficients were slightly under-predicted, but the tendency of relative change of propulsive coefficients for different rudder locations was captured. Finally he concluded that the method was capable of the optimisation of hull/propeller/rudder interaction

Alin et al. computed fully appended submarine-propeller configuration (Alin et al. 2010). The LES computations were performed with the Open-FOAM code in order to investigate propeller hull interaction, effect of a real wake on the propeller compared to open water, flow induced noise and its origin, and variation of force distribution on the propeller during rotation. For the work they developed a Deformation and Regeneration method for unsteady CFD computations which have moving parts like propeller. The method was found to be computationally very effective with only 10 % overhead compared to the fixed grid simulations. The results showed that their computations predicted flow around propelled submarine well but for further validation they mentioned the lack of high quality public experimental data for submarines.

Muscari et al. computed the flow around a propeller behind a fully appended twin screw hull using the in-house unsteady RANS code Xnavis (Muscari et al 2010). In the simulations the real propeller geometry was used with a dynamic overlapping grids approach. The computations were compared against LDV measurements in model scale. Based on visualizations they presented that the main features of the flow were correctly captured. The comparison of the computed results against the experimental data of longitudinal velocity and transverse vorticity in the vicinity of the propeller and rudder showed good results.

### 5.5 Manoeuvring

Numerical simulation of ship manoeuvres at model or full scale is a challenge, due to both the complexity of the physical phenomena involved and the level of capability and resources needed to perform the computations. CFD has been exercised extensively for static problems, including resistance computations, forward-speed diffraction and static manoeuvres (Larsson et al. 2000, Hino 2005) and results for these applications are today mostly deemed accurate for engineering purposes. In the SIMMAN 2008 manoeuvring workshop (Stern and Agdrup 2008, Simman 2008) and elsewhere (Carrica et al. 2006, Atsavapranee et al. 2010, Bhushan et al. 2010) CFD computations of static manoeuvres were presented for pure drift and steady turn. These static “manoeuvres” are used in lieu of experiments to obtain coefficients used in system-based models to predict actual dynamic manoeuvres.

Rotating arm computations can also be performed to obtain manoeuvring derivatives. In this case the computation simulates a steady turn with drift, as would happen on a free sailing ship. The computations can be performed on the ship system of reference, adding body forces to account for the non-inertial accelerations, or on the earth inertial reference system, and then moving all grids following the trajectory of the ship. Either way results in a static computation.

Planar motion mechanism (PMM) computations mimic PMM experiments and provide a wider range of derivatives needed in system-based methods than those provided by static computations. Wilson et al. and Di Mascio et al. performed PMM computations using captive models, i.e. imposing surge, sway and yaw but restricting pitch, heave and roll, and Sakamoto et al. also predicted the resulting pitch, heave and roll (Wilson et al. 2007, Di Mascio et al., 2007, Sakamoto et al. 2008). The main conclusion from SIMMAN 2008 is that CFD methods are mature enough to obtain derivatives needed by system-based methods to simulate ship manoeuvres.

In most captive model computations, however, full 6 degrees of freedom (6DOF) motions and independent moving rudders are not needed. Direct simulation of ship manoeuvres requires self-propulsion, moving rudders and in general full 6DOF capabilities in a free surface environment. There have been only a few computations of this type. Jensen et al. computed the turning circle manoeuvre of a container ship with a body force model for the propeller, resolving the free surface (Jensen et al., 2004). Pankajakshan et al. performed calculations of the ONR Body 1 with active control surfaces and a resolved rotating propeller, with excellent results (Pankajakshan, 2002). Venkatesan and Clark also simulated the ONR Body 1 with 6DOF using an explicit rotating propeller (Venkatesen and Clarke, 2007). These two authors used sliding or deformable grids to compute relative motions between grids, but neglected the free surface.

In particular, Pankajakshan et al. use a series of pre-computed grids at several rotations of the propeller that are later used to interpolate the final grid for an arbitrary propeller rotational angle.

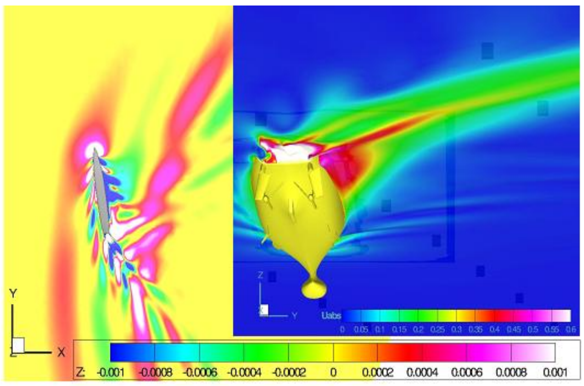
Muscari et al. computed the very large crude carrier model KVLCC2 in a turning-circle manoeuvre using RANS but did not compute the free surface hence limited to only three degrees of freedom (Muscari et al., 2008). Durante et al. simulated a turning manoeuvre adding free surface and 6DoF to a fully appended tanker geometry (Durante, 2010). Carrica and Stern performed turn and zig-zag manoeuvring computations for KVLCC1 with discretized propeller, 6DOF, free-surface and DES, but the KVLCC1's horn rudder was approximated by a spade rudder to simplify the geometry, resulting in over-predicted steering (Carrica and Stern, 2008b). Zig-zag and turning manoeuvres were simulated by Carrica et al. for the fully-appended surface combatant model MARIN 7967 in model and full scale, showing very good agreement with data (Carrica et al. 2008c).

In general free-model manoeuvres are complex and resource intensive, but the results are satisfactory for general variables such yaw rate, yaw, tactical diameter, etc. Experimental data for CFD validation of ship manoeuvring are very limited with some of the earliest being provided by Crane (1979). Recently, however, in SIMMAN 2008 a collection of model scale experimental data are documented for surface combatant (DTMB 5415), very large crude carriers (KVLCC1 and KVLCC2) and containership (KCS) geometries, for both planar motion mechanism (PMM) and free model manoeuvres (Stern and Agdrup 2008).

The study of the flow physics of different phenomena related to manoeuvring, such as propeller-rudder-hull interaction, has been performed using CFD tools for a time, with good results. Performance and efficiency of the control surfaces are studied using CFD in simplified situations (see for instance Guo et al. 2010), providing also information for system-based simulators.

Available data sets for PMM and static manoeuvres are more complete than those for free models, and include forces, moments, motions and flow fields. Though trajectories, motions and forces are available for some cases, fluid flow data for free model manoeuvres is essentially inexistent and is needed to validate CFD tools and procedures before full confidence can be bestowed on these complex computations.

Computational times are highly dependent on the grid size, running platform and CFD package used, but a rule of thumb can be provided for practical grids of a few million grid points running on a modern cluster with 150,000 grid points per processor. Static computations (pure drift, rotating arm) can take 12~36 hours, PMM computations 24~60 hours and free running models 48~168 hours.




.

Fig 5.2. 7967 in turning manoeuvre hit by its own Kelvin wake (Carrica et al. 2008c)

## 5.6 Seakeeping

Computation of seakeeping problems is quickly becoming more popular as more codes add the necessary capabilities to perform these simulations. These capabilities include modelling of waves, regular, or irregular short- or long-crested, and capability to perform predicted motions. Tremendous progress has been made since 2005, when the Seakeeping

Committee if the ITTC stated in their Final Report and Recommendations (ITTC 2005) "seakeeping computations are still far from a state of mature engineering science." From just one case concerning seakeeping (forward speed diffraction which involves no motions) in the Tokyo 2005 CFD Workshop (Hino 2005), several cases involving waves and pitch and heave or pitch, heave and surge were included in the Gothenburg 2010 CFD Workshop (G2010), with numerous contributions for each case.

The first CFD computations of pitch and heave were presented by Sato et al. (1999), with a few contributions before 2008 (Cura Hochbaum and Vogt 2002, Orihara and Miyata 2003, Klemt 2005, Weymouth 2005, Carrica et al. 2007). Many more results have been published since 2008, and most can be found in G2010 and the references in the papers therein. Computation of diverse geometries including containerships, surface combatants, tankers, high-speed craft, etc. has been demonstrated. Grids as coarse as 400,000 points (Sato, 1999) and as fine as 70 M points (Carrica et al., 2010a) have been used, with a clear trend towards increasingly better results as the grids are finer. While the computed amplitude of the motions is generally well within 10% of the experiments, the prediction of the added resistance is more difficult (G2010, 2010).

G2010 has seen 4 contributions for forward speed diffraction computations using the codes Surf, CFDShip-Iowa v4.5, Icare and Fluent, comparing against free surface elevation, flow field at the nominal wake plane and forces and moment. Pitch and heave computations were contributed by 5 groups for KCS using FreSCo, Comet, Open Foam, Wisdam and CFDShip-Iowa v4.5 for KCS in head waves under three different conditions. Comparisons are performed against motions and transient resistance. In all cases CFD performs well, with the exception of the resistance for pitch and heave, for which the experimental data is unreliable (G2010, 2010). Five groups also contributed with pitch and heave computations of the tanker KVLCC2, running Comet, Open Foam, CFDShip-Iowa V4.5, Isis, Icare and RIAM-CMEN, comparing against forces and motions.

Computations of pitch and heave free to surge are scarce, but a case with three wavelengths was included in G2010 for the KVLCC2 tanker, which attracted two submissions (el Moctar et al. 2010 using Comet, Sadat-Hosseini et al. 2010 using CFDShip-Iowa v4.5). The reported average errors on 1st harmonic amplitudes and phases for motions are 9.32% and 14.5%, respectively, but for forces increase significantly. CFD predicted well the decrease in forces when the ship is free to surge.

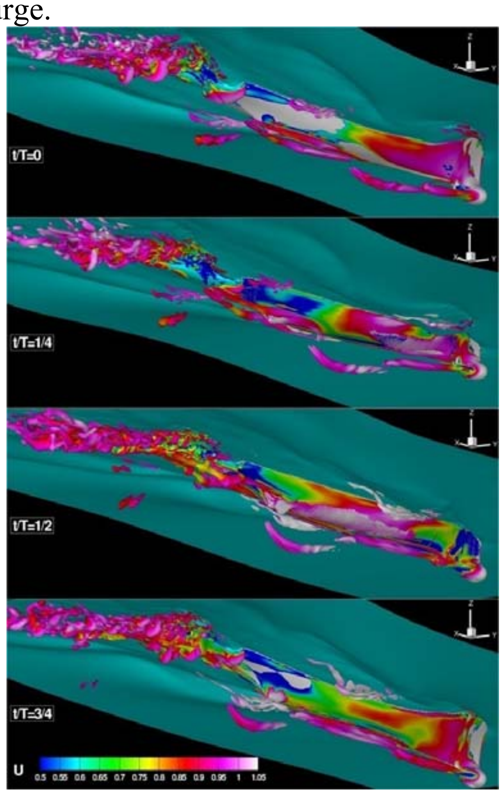



Fig 5.3. Turbulent structures colored by axial velocity at the four quarter periods for DTMB 5415 pitching and heaving in regular head waves (Carrica et al. 2010a).

Pitch and heave computations of RAOs can be expensive due to the large number of runs needed for every Froude number at different encounter frequencies. Mousaviraad et al. (2010) demonstrate procedures that compute the RAOs for a Froude number in a single run, greatly reducing the computation time. This methodology has essentially the same order of errors found in standard single wave methods if the response is linear (small waves).

Full 6DoF computations of seakeeping are more complex because require use of a propeller to achieve self-propulsion. Examples of these types of computations for the problem of broaching of a surface combatant are presented by Carrica et al. and Sadat-Hosseini et al., this last comparing against experimental data with good agreement. These computations are highly non-linear and very resource intensive and are currently limited to academic research and special applications (Carrica et al. 2008a and Sadat-Hosseini et al. 2009).

Since roll decay is dominated by viscous effects, potential flow codes are largely ineffective to treat these problems. CFD computations have been performed initially using deformable grids (Wilson et al. 2006 computed the DTMB 5415 surface combatant with bilge keels), but surface capturing approaches are standard today. G2010 evaluates the same roll decay case as Wilson et al. for 4 submissions, showing average differences of roll angle with experiments of 0.85%, but 10% for forces. Commercial (Fluent), in house (Isis and Icare) and open source codes were used.

Validation data for seakeeping including motions, forces and moments, and flow field is still needed. In particular, no flow field information is available for pitch and heave experiments, and usually the forces and moments are unreliable. Though experiments measuring transient flows around ships in waves are complex, these are necessary to boost confidence on CFD predictions.

### 5.7 Ocean Engineering

There are a great variety of CFD applications in ocean engineering area and significant developments can be found in recent years. The focuses are placed on problems of non-linearity, viscosity and fluid-structure interaction (FSI). In this section, practical applications and generic concerns in ocean engineering fields as shown in following are reviewed.

a) Coupled wind-wave and wind loads simulation
b) Wave/fluid-structure interactions, including viscous effects
c) Violent Flows, slamming, sloshing, green water on deck, impact
d) Cylinder flows, VIV, risers

On the other hand, the numerical methods and schemes themselves have been also significantly developed in the past few years.

a) Hybrid methods for potential/viscous flow coupling
b) Verification, validation and uncertainty analysis

<u>Coupled wind-wave and wind loads simulation.</u>

The simulation of wind-wave interaction and other complicated marine environment aiming at the prediction of external forces acting on marine structure is a new area in which CFD has showed significant progress in recent year. Shen et al. employed a high-order spectral method (HOS) to capture the non-linear processes in realistic ocean wave simulation (Shen et al. 2008), while the turbulent wind motions were computed by LES with the precise sea surface geometry. Wind loads and wave loads had been possibly identified by taking the two-way interaction in large scale and realistic wind-wave field into account. Koop et al. calculated wind loads on a tandem off-loading configuration using CFD (Koop et al. 2010), and showed that the use of

CFD for estimating wind loads, wing wakes and shielding effects can be feasible by improving the grid and the volumetric refinement box through comparisons with experiments in wind tunnel.

Wave/fluid-structure interactions with viscous effects.

Wave-structure interactions including viscous effects and/or extreme waves are still a challenging problem of ocean engineering. Some recent developments of CFD tools as well as numerical techniques for computing wave-structure interactions in non-linear/breaking waves are attempted. Monroy et al. presented recent advances of the SWENSE (Spectral Wave Explicit Navier-Stokes Equations) approach, a method for simulating fully non-linear wave-structure interactions including viscous effects (Monroy 2009). Baso et al. presented a coupled Eulerian scheme with two Lagrangian particles, ie. SPH and free surface particle on Eulerian grids and applied it to the evaluation of the motion in non-linear waves (Baso et al. 2010).

Violent Flows, slamming, sloshing, green water on deck, impact.

CFD is clearly a powerful tool for simulating the violent flows related to slamming, sloshing, green water on deck and impact. Although there are vast methods, it is still expected that the development with the CFD field will gradually lead to advanced tool that are robust enough for engineering prediction. Yang et al. applied 3D CIP (constrained Interpolation Profile) method to solve Navier-Stokes Equations for water entry problems of 3D bodies (Yang et al. 2010). Large deforming free surface is captured and numerical results of slamming forces are compared with experiments. Hu et al. compared CIP scheme and RCIP scheme for predicting the violent sloshing flow in a 2-D rectangular tank and showed that RCIP scheme has better accuracy than CIP scheme for both free surface and impact pressure (Hu et al. 2010). Lu et al. proposed a numerical time domain model to predict green water impact on fixed and moving FPSO model, in which a VOF (Volume-of-Fluid) technique is used to capture the violent free surface by solving equations using projection scheme and a finite element method on unstructured grids (Lu et al. 2010).

The modified VOF has been considered to be an efficient scheme for solving NS equations for sloshing problem. Wemmenhove et al. and Liu et al. applied improved VOF and Young’s VOF for 2D tank section and LNG-FPSO tank respectively (Wemmenhove et al. 2009 and Liu et al. 2010). Chen et al. studied the sloshing in containers by solving the Reynolds-Averaged NS (RANS) equations based on a two-phase, compressible fluid flow model and captured free surface by an implicit level-set scheme (Chen et al. 2009a, 2009b). It has been shown that air compressibility has a significant effect on the behaviour of wave impacting pressure.

Cylinder flows, VIV, risers.

There are vast applications associated with cylinder flow problem in ocean engineering. The biggest concern nowadays might be vortex induced vibration (VIV) problem. Since the specialist committee on VIV will give practical explanation and guideline on VIV prediction including CFD approaches, we omit it here due to lacking of space.

Hybrid methods for potential/viscous flow coupling.

In order to overcome the difficulties associated with cumbersome grid requirements, an efficient VOF based RANS method is proposed. (Woeckner et al. 2010) In which, the viscous RANS is implicitly forced to comply with a prescribed solution towards the far-field boundaries, for the aim to accurately predict the wave propagation towards floating object and concurrently suppress wave reflections at the outlet boundary. Reported examples indicate that the procedure provides a fair predictive accuracy at low computational coasts. Hybrid method had been widely used for the analysis of coupling problem, such as sloshing coupled with ship motion, in which the internal flow is solved by CFD method while the external flow is calculated based on non-viscous method, to save the computation burden. Kim Y applied the finite difference method (FDM) and smoothed particle hydrodynamics (SPH) to simulate the violent sloshing flow for global and local fluid motion respectively, and then coupled them with an impulsive response function (IRF) for linear and non-linear ship motion (Kim Y, 2007). Wang et al. coupled a finite difference method for sloshing flow with linear trip method and simulated the both ship motions and sloshing motion simultaneously in time domain (Wang et al., 2010).

<u>Verification, validation and uncertainty analysis.</u>

Increasing demands for the development of offshore exploration in harsh environmental area require sufficiently accurate and practical design tools. Despite numerous numerical methods, few methods are applicable for actual engineering use such as the simulation of violent sloshing flow and the prediction of impact loads. Recently some examples of validations related to numerical prediction of sloshing with experimental data including full-scale data can be found, and among these, the Sloshel Project (see Brosset et al. 2009, Maguire et al. 2009) aiming to reproduce at full-scale the wave impact condition due to sloshing and to build a database through the measurements of fluid dynamics as well as structural response is worth mentioning. Besides, Zheng et al. (2009) summarized LR's recent works on validations of different numerical codes and modelling methodologies, such as uncoupled, 1-way and 2-way coupled fluid-structure interaction (FSI) for sloshing at low filling level by using model and full scale test data. Eca et al. proposed a procedure for CFD code verification based on the method of the Manufactured Solution (Eca et al. 2010). Finally, validation is an on-going activity that intends to estimate the modelling error of a given mathematical model. It requires comparisons with experimental data (physical models) and it involves numerical, experimental and parameter uncertainties. A procedure combining all these quantities has been proposed recently by the ASME (see Roache, 2009 and ASME Committee, 2010).

### 5.8 Simulation Based Design Optimization

Under the pressure of technological competition, and thanks to the development of strong and efficient techniques for several practical applications, simulation is today an important part of new design evaluation. Simulation Based Design (SBD) optimization methods integrate simulation codes with minimization algorithms, seeking the best design to perform the desired tasks. The use of SBD optimization tools is rapidly growing in most of engineering disciplines. Naval and off-shore engineering follow at a somewhat slower pace, partially because SBD optimization requires a large set of basic computer tools. This is the case of several CAD tools for geometric modelling, engineering analysis methods, gradient-based or gradient-free optimization algorithms.

Developments and applications to ship design problems are reported in a number of papers (for a recent overview of methods and applications see Campana et al. 2009a). Kim et al. investigate on the flexibility of use of some choices of design variables (local and global) in the optimization of the KCS containership (Kim et al. 2010). Kuhn et al. focussed on the competition of different objective functions in a multi-objective problem (Kuhn et al. 2010).

Computationally expensive problems have been solved in Tahara et al. for a fast multi-hull ship, with particular emphasis on dealing with complex and realistic geometrical and functional constraints (Tahara et al. 2008). In Peri and Campana the use of the Variable Fidelity is combined with a Trust Region approach to combine in a systematic way a High Fidelity RANS solver and a Low Fidelity potential flow code, leading to a substantial reduction of the CPU time without losing the accuracy of the original high fidelity problem (Peri and Campana, 2008). Two new derivative free global optimization algorithms have been presented and tested in Campana et al. (2009b). The test included well known existing global optimization algorithms. The problem solved was that of the improvement of the seakeeping characteristics of a containership.

Deterministic optimum designs are pushed to design constraint boundaries. Robust and Reliability Based Design Optimization methods (RDO and RBDO, respectively) are probabilistic algorithms for quantifying the effect of uncertainties on response metrics of interest and, in conjunction with simulation software, may be employed for designing systems subject to probabilistic performance criteria. RDO is defined in terms of the first two moments of the original objective function and the constraints are given in terms of deterministic inequalities constraints (for a recent RDO review see Park et al., 2007). The RBDO problem is the task of handling the constraints defined in terms of probabilistic inequality, during the minimization procedure.

RDO and RBDO applications are also starting to appear in the field of ship design. An RDO formulation is used in Kramer et al. for a first-order design of a waterjet propulsion system for a full-scale SES ship (Kramer et al., 2010). Diez et al. developed a complete formulation for the Multidisciplinary Robust Design and presented preliminary results for conceptual design problems and for a sailing yacht keel hydro-elastic optimal design (Diez et al., 2010).

### 5.9 Conclusions

Due to improvements in code capabilities and computer performance and access, CFD use in practically all areas of naval architecture has increased dramatically over the past decade. If properly performed, resistance and propulsion computations are within acceptable uncertainties for most applications, as well as computations of propulsor performance. Though it requires more computational resources and advanced code capabilities, CFD use for manoeuvring and seakeeping is becoming more commonplace, though validation of the CFD results and procedures has been more limited. The same trend is observed for ocean engineering applications and shape optimization, the latter very resource intensive. The continuing trend of enormous improvements in computer performance, numerical methods and turbulence modelling, along with a decrease of the cost of access, anticipate an ever wider use and acceptance of CFD as a naval architecture tool for more everyday as well as more complex applications.

Limitations on validation exist due to a lack of good quality data for full-scale problems, transient turbulence including unsteady frequency content, local forces on appendages, and detailed flow field data

## 6. RECOMMENDATIONS

Adopt the procedure No. 7.5-03-02-03 Practical Guidelines for Ship CFD Application.

## 7. REFERENCES

Abdel-Maksoud M (Ed), 2011,"Proceeding of Second International Symposium on Marine Propulsors", Hamburg, Germany

Abdel-Maksoud M, Steden M,Hundemer J,2010, "Design of a Multi-Component Propulsor", ONR 2010

Alessandrini B., Delhommeau, G. 1999. "A FullyCoupled Navier-Stokes Solver for Calculations ofTurbulent Incompressible Free Surface Flow Past AShip Hull", Intern. J. Num. Methods in Fluid, Vol. 29, pp. 125-142.

Alin N., Chapuis M., Fureby C., Liefvendahl M., Svennberg U., Troëng C., 2010, "A Numerical Study of Submarine Propeller-Hull Interactions", 28th Symposium on Naval Hydrodynamics, Pasadena, California.

Aren P and Lundberg J, 2008, "Rolls-Royce combines CFD and Cavitation Tunnel Testing in Development of New Kamewa CP-A" , Marine CFD 08 Conference.

ASME Committee PTC-61(2008), 2010. ANSI Standard V&V 20. "ASME Guide on Verification and Validation in Computational Fluid Dynamics and Heat Transfer"

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

Baso S., Mutsuda H., Kurihara T., Kurokawa T., Doi Y., Shi J., 2011, "An Eulerian Scheme with Lagrangian Particle for Evaluation of Seakeeping Performance of Ship in Nonlinear Wave", Intern. J. Offshore and Polar Engineering, Vol.21, No.2, pp.103-110.

Bensow R.E. and Bark G.. 2010, "Implicit LES Predictions of the Cavitating Flow on a Propeller", J. Fluids Engineering. Vol.132, Issue 4, 041302 1-10.

Bhushan S, Carrica P, Yang J, Stern F, 2010. "Large-Scale Parallel Computing and Scalability Study for Surface Combatant Static Maneuver and Straight Ahead Conditions Using CFDShip-Iowa." J. High Perf. Comput., in press.

Brossel L, Mravak Z, Kaminski M, Collins S, Finnigan T, 2009. "Overview of Sloshel Project", 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Bulten N, 2008, "Determination of Transient Shaft Forces in Waterjets and Thrusters based on CFD Analysis", RINA 2008

Bulten N, Nijland M, 2011, "On the Development of a Full-Scale Numerical Towing Tank: Reynolds Scaling Effects on Ducted Propellers and Wakefields", SMP 11

Califano A, Steen S, 2009, "Analysis of different propeller ventilation mechanisms by means of RANS simulations", SMP09

Campana EF, Peri D, Tahara Y, Kandasamy M, Stern F, 2009a."Numerical Optimization Methods for Ship Hydrodynamic Design," SNAME Annual Meeting, Providence (Rhode Island, USA).

Campana EF, Peri D, Lucidi S, Pinto A, Liuzzi G, Picciallli V, 2009b. "New Global Optimization Methods for Ship Design Problems", Optimization and Engineering , Vol. 10 (4), 533.

Carlton J, Radosavljevic D, Whitworth S, 2009, "Rudder – Propeller – Hull Interaction: The Results of Some Recent Research, In-Service Problems and Their Solutions", SMP09

Carrica PM, Huang J, Noack R, Kaushik D, Smith B, Stern F, 2010a. "Large-Scale DES Computations of the Forward Speed Diffraction and Pitch and Heave Problems for a Surface Combatant". Comput. Fluids39, 1095-1111.

Carrica PM, Castro A, Stern F, 2010b. "Self-Propulsion Computations Using Speed

Controller and Discretized Propeller with Dynamic Overset Grids", J. Marine Sci. Tech. 15, 316-330.

Carrica PM, Paik K, Hosseini H, Stern F, 2008a. "URANS Analysis of a Broaching Event in Irregular Quartering Seas", J. Marine Sci. Tech.13, 395-407.

Carrica PM, Stern F, 2008b. "DES Simulations of KVLCC1 in Turn and Zigzag Manoeuvres with Moving Propeller and Rudder". SIMMAN 2008, Copenhagen, Denmark.

Carrica PM, Ismail F, Hyman M, Bhushan S, Stern F, 2008c. "Turn and Zigzag Manoeuvres of a Surface Combatant Using a URANS Approach with Dynamic Overset Grids". SIMMAN2008, Copenhagen, Denmark.

Carrica PM, Wilson RV, Noack R, Stern F, 2007. "Ship Motions using Single-Phase Level Set with Dynamic Overset Grids", Comput. Fluids 36, 1415-1433.

Carrica PM, Wilson RV, Noack R, Xing T, Kandasamy M, Stern F, 2006. "A dynamic overset, single-phase level set approach for viscous ship flows and large amplitude motions and maneuvering". 26th ONR Symp. Naval Hydrodyn., Rome, Italy.

Chapple M and Renilson M, 2009, "A viable approach to propeller safety for small craft:Ringed Propellers", SMP09

Chen Y G, Djidjeli K, Price W G, 2009a. "Numerical Simulation of Liquid Sloshing Phenomena in Partially Filled Container". Computers & Fluids 38.

Chen Y G, Price W G, Temarel, T, 2009. "Numerical Simulation of Liquid Sloshing in LNG Tanks Using a Compressible Two-Fluid Flow Model". 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Chesnakas CJ, Donnelly MJ, Pfitsch DW, Becnel AJ andSchroeder SD, 2010,"Pump Loop Testing of an Axial Flow Waterjet", ONR 2010

Choi J.E., Kim J.H., Lee H.G, Choi B.J., Lee D.H., 2009, "Computational predictions of ship-speed performance", Journal of Marine Science and Technology 14(3), 322–333.

Choi J.E., Min K.-S., Kim J.H., Lee S.B., Seo H.W., 2010, "Resistance and propulsion characteristics of various commercial ships based on CFD results," Ocean Engineering 37, 549–566.

Cointe R, Tulin MP, 1994," A theory of steady breakers". J FluidMech 276:1.

Coutier-Delgosha, O. Fortes-Patella, R. and Reboud, J.L., 2002 "Simulation of unsteady cavitation with a two-equation turbulence model including compressibility effects", Journal of Turbulence, 3 (2002) 058.

Crane CL, 1979. "Maneuvering trials of 278000-DWT tanker in shallow and deep waters". Trans. SNAME87, 251-283.

Cura Hochbaum A, Vogt M, 2002. "Towards the Simulation of Seakeeping and Maneuvering based on the Computation of the Free Surface Viscous Flow". 24th ONR Symp Naval Hydrodyn., Fukuoka, Japan.

Delannoy, Y. and Kueny, J.L., 1990, "Two Phase Flow Approach in Unsteady Cavitation Modelling", Proc. Cavitation and Multiphase Flow Forum, ASME-FED, Vol. 98.

Di Mascio A, Broglia R, Muscari R, 2007. Numerical Simulations of Flow Around a Fully Appended Hull with Enforced Motion. 9th Int. Conf. Num. Ship Hydrodyn., Ann Arbor, MI.

Diez M, Peri D, Fasano G, Campana EF, 2010. Multidisciplinary Robust Optimization for

Ship Design, 28th Symp. Naval Hydrodyn., Pasadena (CA), USA.

Dommermuth D, 2010. Personal comm..

Dular, M., Bachert, R., and Stoffel, B., 2006 "Experimental and numerical investigation of swept leading edge influence on the developed cavitation", 6thInt'l Symposium on Cavitation (CAV2006), The Netherlands.

Durante D, Broglia R, Muscari R, Di Mascio A, 2010. Numerical Simulations of a Turning Circle Maneuver for a Fully Appended Hull. 28th Symp. Naval Hydrodyn., Pasadena, CA.

Eça, L., Hoekstra, M., 2006, "Discretization Uncertainty Estimation Based on a Least Squares Version of the Grid Convergence Index," Proc. II Workshop on CFD Uncertainty Analysis, IST, Lisbon (Portugal).

Ferrant, P, Gentaz, L., Monroy C., Luquet, R. Ducrozet, G, Alessandrini, B., Jacquin E., Drouet, A, 2008. Recent Advances Towards the Viscous Flow Simulationof Ships Manoeuvering in Waves, 23rd International Workshop on Water Waves and Floating Bodies .

Funeno I, 2009, "Hydrodynamic Optimal Design of Ducted Azimuth Thrusters", SMP09

Gaggero S, Brizzolara, 2009, "Parametric optimisation of fast marine propellers via CFD calculation." FAST 2009, Athens, Greece

Gicquel LYM, Staffelbach G, Cuenot B, Poinsot T, 2008. "Large Eddy Simulations of Turbulent Reacting Flows in Real Burners: the Status and Challenges". J. Phys.: Conf. Ser. 125 012029.

Go S, Seo H, Choi G, 2009, "Study on the Powering Performance Evaluation for the Pod Propulsion Ships", SMP09

Guo C, Hu W, Huang S, 2010. "Using RANS to simulate the interaction and overall performance of propellers and rudders with thrust fins". J. Arine Sci. Appl. 9, 323-327.

Han K.-J., 2008, "Numerical optimization of hull/propeller/rudder configurations", PhD thesis, Chalmers University of Technology.

Hinatsu, M, Tsukada, Y. Fukasawa, R and Tanaka, Y., 2001. "Experiments of Two-phase Flows for the Joint Research, Proc. SRI-TUHH mini Workshop on NumericalSimulation of Two-Phase Flows"

Hino T (Editor), 2005. CFD Workshop Tokyo 2005. National Maritime Res. Inst., Japan.

Hino, T. Ohashi, K and Kobayashi, H. 2010, "Flow Simulations Using Navier-Stokes Solver Surf", Proc. of G2010 Workshop.

Hirata N, Hino T, 1999, "An Efficient Algorithm For SimulatingFree-Surface Turbulent Flows Around an Advancing Ship". J Soc NavArchit Jpn 185:1

Hu C, Yang K, Kim Y, 2010. "3-D Numerical Simulations of Violent Sloshing by CIP-based Method". 9th Intl. Conf. on Hydrodyn. Shanghai, China.

Hu, C. and Kashiwagi, M. 2010, "CIP Based Cartesian Grid Method for Prediction of Nonlinear Ship Motions", Proc. of G2010 Workshop

Iafrati, A., Di Mascio, A. and Campana, E.F. 2001, "A Level Set Technique Applied to Unsteady Free Surface Flows", Int. Journal for Numerical Methods in Fluids, Vol.35.

ITTC, 2005, "Seakeeping Committee Report", Proc. 24th Int. Towing Tank Conf., Edinburgh, Scotland.

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

Jang H, Mahesh K,2010, "Large Eddy Simulation of Marine Propulsors in Crashback", ONR 2010

Jensen G, Klemt M, Xing Y, 2004. "On the way to the numerical basing for seakeeping and maneuvering". 9th Sym. Practical Design of Ships, Luebeck, Germany.

Kandasamy M, Takai T, Stern F, 2009, "Validation of detailed water-jet simulation using URANS for large high-speed sea-lifts", FAST 2009

Kim H, Yang Chi, Kim HJ and Chun HH, 2010. "A Combined Local and Global Hull Form Modification Approach for Hydrodynamic Optimization", 28th Symp. Naval Hydrodyn., Pasadena (CA), USA.

Kim S.E and Brewton S., 2008, "A Multiphase Approach to Turbulent Cavitating Flows", The 27th Symposium of Naval Hydrodynamics, Seoul, Korea.

Kim S-E and Schroeder S, 2010, "Numerical study of thrust breakdown due to Cavitation on a Hydrofoil, a Propeller and a WaterJet", ONR 2010

Kim Y, 2007. "Experimental and Numerical Analyses of Sloshing Flows". J Eng Math 58:191–210.

Kim, J., Park, I-R., Kim, K-S. and Van, S-H. 2010, "Feasibility Study on Numerical Towing Tank Application to Predictions of Resistance and Self-Propulsion Performances for a Ship", Proc. of G2010 Workshop

Kinnas SA, Chang S-H, and Yu Y-H, 2010, "Prediction of Wetted and Cavitating Performance of Water-jets", ONR 2010

Klemt M, 2005. "RANSE Simulation of Ship Seakeeping using Overlapping Grids". Ship Tech. Res. 52, 65-81.

Koop A, Klaij C, Vaz G, "Predicting Wind Loads for FPSO Tandem Offloading Using CFD", 29th Intl. Conf. Ocean, Offshore and Arctic Eng. (OMAE). Shanghai, China.

Koushan K, Steen S (Editors), 2009, "Proceedings of the First International Symposium on Marine Propulsion", Trondheim, Norway

Kramer M, Motley M, Young Y-L 2010. "Probabilistic-Based Design of Waterjet Propulsors for Surface Effect Ships", 29th American Towing Tank Conference, Annapolis (MD), USA.

Krasilnikov V, Sun J and Henning Halse H, 2009, "CFD Investigation in Scale Effect on Propellers with Different Magnitude of Skew in Turbulent Flow" 1st Symp. on Marine Propulsion, Trondheim, Norway

Krasilnikov V, Zhang Z and Hong F, 2009, "Analysis of Unsteady Propeller Blade Forces by RANS", SMP09

Kremenetsky M, 2008. "Considerations for Parallel CFD Enhancements on SGI NUMA and Cluster Architectures". http://www.fluent.com/software/fluent/fl5bench/otherart/oart1.htm.

Kuhn J, Collette M, Lin WM, Schlageter E, Whipple D, Wyatt D, 2010. "Investigation of Competition Between Objectives in Multiobjective Optimization", 28th Symp. Naval Hydrodyn., Pasadena (CA), USA.

Kunz, R.F., Boger, D.A. and Stinebring, D.R., 2000, "A Pre-conditioned Navier-Stokes Method for Two-Phase Flows with Application to Cavitation Prediction", Computers & Fluids, Vol. 29.

Larsson L, Stern F, Bertram V, 2003. "Benchmarking of computational fluid dynamics for ship flows: The Gothenburg

2000 workshop". J. Ship Res. 47, 63-81.

Larsson L., Zou L., 2010b, "Gothenburg 2010 – Evaluation of resistance, sinkage and trim, self propulsion and wave pattern predictions".

Larsson, L. Stern F. Visonneau, M (Editors), 2010, Proceedingsof G2010Workshop on NumericalShip Hydrodynamics.

Li, D-Q and Grekula M., 2008, "Prediction of dynamic shedding of cloud cavitation on a 3D twisted foil and comparison with experiments", The 27thSymposium of Naval Hydrodynamics, Seoul, Korea.

Liefvendahl M, Felli M,Troëng C, 2010,"Investigation of Wake Dynamics of a Submarine Propeller", ONR 2010

Lifante C, Frank T and Reich K, 2008, "Investigation of Pressure Fluctuations caused by Turbulent and Cavitating Flow around a P1356 Ship Propeller", RINA'08

Lindau JW, Moody WL, Kinzel MP, Dreyer JJ, Kunz RF, Paterson EG, 2009, "Computation of Cavitating Flow through Marine Propulsors", SMP09

Liu Y T, Ma N, Gu X C, 2010. "Numerical Simulation of Large Amplitude Sloshing in FPSO tanks under Irregular Excitations Based on Young VOF Method". 29th Intl. Conf. Ocean, Offshore and Arctic Eng.. (OMAE), Shanghai, China.

Lu H, Yang C, Lohner R, 2010. "Numerical Studies of Green Water Impact on Fixed and Moving Bodies". 20th Intl. Offshore and Polar Eng. Conf. (ISOPE), Beijing, China.

Maguire J R, Whitworth S, Oguibe CN, Radosavljevic D, Carden EP, 2009. "Sloshing Dynamics – Numerical Simulations in Support of the Sloshel Project". 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Manzke, M. and Rung, T., 2010,"Resistance Prediction and Sea Keeping Analysis with FreSCo+", Proc. of G2010 Workshop.

Merkle, C.L., Feng, J., and Buelow, P.E.O., 1998, "Computational Modeling of the Dynamics of Sheet Cavitation", Proc. 3rd Intl. Symposium on Cavitation, Grenoble, France.

Minchev A, Nielsen JR, Lundgren E, 2009, "Ducted Propeller Design and Verification for Contemporary Offshore Support Vessels", SMP09

Monroy C, Ducrozet G, Roux de Reilhac P, Ferrant P, Alessandrini B, 2009. "RANS Simulation of Ship Motion in Regular and Irregular Head Seas Using the SWENSE Method". 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Morgut M, Nobile E, 2009, "Comparison of Hexa-Structured and Hybrid-Unstructured Meshing Approaches for Numerical Prediction of the Flow Around Marine Propellers", SMP09

Muller S-B, Abdel Maksouk M, Hilber G, 2009,"Scale effects on propellers for large container vessels", 1st Symp. on Marine Propulsion, Trondheim, Norway

Muscari R, Broglia R, Di Mascio A, 2008. "Trajectory prediction of a self-propelled hull by unsteady RANS computations". 27thSymp. Naval Hydrodyn., Seoul, Korea.

Muscari R, Di Mascio A, 2009, "Simulation of the viscous flow around a propeller using a dynamic overlapping grid approach", SMP09

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

Naval Hydrodynamics, Pasadena, California.

Muscari, R. and Di Mascio, A. 2004. "Numerical modeling of breaking waves generated by a ship's hull" J Mar Sci Technol (2004) 9:158

O'Shea T, Brucker K, Dommermuth D, Wyatt D, 2008. "A Numerical Formulation for Simulating Free-Surface Hydrodynamics", 27th Symp. Naval Hydrodyn., Seoul, Korea.

Ochi F, Fujisawa T, Ohmori T, Kawamura T, 2009, "Simulation of propeller hub vortex flow", SMP09

Oger G, Rousset J.M. Le Touze D. Alessandrini B Fxerrant P, 2007, "SPH simulations of 3-D slamming problems", Proc. 9th Intern. Conf. Num.l Ship Hydro Ann Arbor.

Orihara H, Miyata H, 2003. "A Numerical Method for Arbitrary Ship Motions in Arbitrary Wave Conditions Using Overlapping Grid System". Proc. 8th Int. Conf. Num. Ship Hydrodyn., Busan, Korea.

Özsu E, Takinaco AC, Odabsi AY, 2009, "Viscous/Potential Flow Coupling Study for Podded Propulsors", SMP09

Palm M, Jürgens D, Bendl D, 2010, "CFD Study on the Propeller-Hull-Interaction of Steerable Thrusters", NuTTS 2010

Palm M, Jürgens D, Bendl D, 2011, "Numerical and Experimental Study on Ventilation for Azimuth Thrusters and Cycloidal Propellers", SMP 11

Pan Y, S. Kinnas SA, 2009 "A Viscous/Inviscid Interactive Approach and its Application to Hydrofoils and Propellers with Non-zero Trailing Edge Thickness", 1st Symp. Marine Propulsion, Trondheim, Norway

Pankajakshan R, Remotigue S, Taylor L, Jiang M, Briley W, Whitfield D, 2002. "Validation of control-surface induced submarine maneuvering simulations using UNCLE". 24th ONR Symp. Naval Hydrodyn., Fukuoka, Japan.

Park GJ, Lee TH, Lee KH, Hwang KH, 2006. "Robust design: an Overview", AIAA Journal Vol. 44, No. 1, 181-191.

Peri D, Campana EF, 2008. "Variable Fidelity and Surrogate Modelling in Simulation-Based Design", 27th Symp. Naval Hydrodyn., Seul (Korea).

Peri D, Campana EF, Tahara Y, Takai T,Kandasamy M, Stern F, 2010, "New developments in Simulation-Based Design with application to High Speed Waterjet Ship Design", ONR 2010

Queutey P, Visonneau M (2007) "An interface capturing method for free-surface hydrodynamic flows". Comput Fluids 36(9):1481–1510

Reboud, J.L., Stutz, B. and Coutier, O., "Two-phase flow structure of cavitation: Experiment and modelling of unsteady effects", Proc. of the 3rd International Symposium on Cavitation, Grenoble, France, 1998.

Rhee SH, Stern F (2002) "RANS model for spilling breaking waves". J Fluid Eng 124:1.

Roache PJ, 2009. "Fundamentals of Verification and Validation". Hermosa Publishers, Albuquerque, New Mexico 2009

Sadat-Hosseini H, Carrica PM, Stern F, Umeda N, Hashimoto H, Yamamura S, Mastuda A, 2009. "Comparison CFD and System-Based Methods and EFD for Surf-riding, Periodic Motion, and Broaching of ONR Tumblehome", 10th Int. Conf. Stability Ships and Ocean Vehicles, St. Petersburg, Russia.

Sakamoto N, Carrica PM, Stern F, 2008. "URANS simulations for static and dynamic maneuvering for a surface combatant". Proc. SIMMAN 2008, Copenhagen, Denmark.

Salvatore F, Streckwall H, van Terwisga T, 2009, "Propeller Cavitation Modelling by CFD – Results from the VIRTUE 2008 ROME Workshop" , Trondheim, Norway

Sasaki N, 2011, "Boundary Layer Control of Twin Skeg Hull Form with Reaction Podded Propulsion", SMP 11

Sato Y, Miyata H, Sato T, 1999. "CFD Simulation of 3D Motion of a Ship in Waves: Application to an Advancing Ship in Regular Heading Waves". J. Marine Sci. Tech. 4, 108-116.

Schmidt, S.J. Sezal, I.H. and Schnerr, G.H. 2006, "Compressible Simulation of High-Speed Hydrodynamics with Phase Change". ECCOMAS CFD.

Schnerr, G. H. and Sauer J., 2001, "Physical and Numerical Modeling of Unsteady Cavitation Dynamics", 4th International Conference on Multiphase Flow, New Orleans, USA.

Schroeder S, Kim S-E, Jasak H, 2009, "Toward Predicting Performance of an Axial Flow Waterjet Including the Effects of Cavitation and Thrust Breakdown", SMP09

Senocak, I. and Shyy, W., 2002, "Evaluation of Cavitation Models for Navier-Stokes Computations", Proc. Cavitation and Multi-Phase Flow Forum. ASME FEDSM 2002, Montreal, CA.

Shen L, Yang D, Yue DKP, 2008. "Coupled Wind-Wave Prediction for Ship Motion". 27th Symp. Naval Hydrodyn., Seoul, Korea.

Shibata, K.Koshizuka, S and Tanizawa, K 2009, "Three-dimensional numerical analysis of shipping water onto a moving ship using a particle method", J. of Marine Sci. and Tech.,Vol.14, 2 , Page 214-227

Sileo L, Steen S, 2009, "Numerical investigation of the interaction between a stern tunnel thruster and two ducted main propellers", SMP09

Simman 2008. http://www.simman2008.dk/

Simonsen CD, Carstens E, 2009, "RANS Simulation of the flow around a ship appended with rudder, ice fins and rotating propeller", NuTTs 2009

Simonsen CD, Nielsen CK and Kishev Z, 2010, "Using CFD for simulation of ships with different fuel saving rudder-propeller devices", NuTTs 2010

Singhal, A.K., Athavale, M.M., Li, H.Y., Jiang, Y., 2002, "Mathematical Basis and Validation of the Full Cavitation Model," J. of Fluids Engineering, Vol.124, pp. 617 624.

Song, C.C.S. and He, J., 1998, "Numerical Simulation of Cavitating Flows by Single-Phase Flow Approach", Proc. 3rd Intl. Symposium on Cavitation, France.

Starke, B, van der Ploeg, A. Raven、H, 2010, "Viscous free surface flow computations for self-propulsion conditions using PARNASSOS", Proc. of G2010 Workshop

Steden M, Hundemer J, Abdel-Maksoud M, 2009, "Optimisation of a Linearjet", SMP09

Steenson LV, Phillips AB, Furlong ME, Rogers E, Turnock SR, 2011, "The Performance of Vertical Tunnel Thrusters on an Autonomous Underwater Vehicle Operating Near the Free Surface in Waves" SMP 11

Stern F, Agdrup K (editors), 2008. "SIMMAN 2008 workshop on verification and validation of ship maneuvering simulation methods". Copenhagen, Denmark.

Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., 2001, "Comprehensive Approach to Verification and Validation of CFD Simulations – Part1: Methodology and Procedures", J. Fluids Eng., 124, pp. 793.

Streckwall H and Salvatore F, 2008, "Results from the Wageningen 2007 Workshop on Propeller Open Water Calculations", RINA Symposium on CFD, London , UK

Tahara Y, Peri D, Campana EF, Kandasamy M, Stern F, 2008. "Single and Multi objective Design Optimization of a Fast Multihull Ship: numerical and experimental results", 27th Symp. Naval Hydrodyn., Seul (Korea).

Venkatesan G, Clark W, 2007. "Submarine Maneuvering Simulations of ONR Body 1". Proc. OMAE2007, San Diego, CA.

Wackers, J, Koren, B... Raven, H.C. van der Ploeg, A, Starke, A.R.Deng, G.B., Queutey, P.,Visonneau, M, Hino, T. and Ohashi K., 2011 "Free-Surface Viscous Flow Solution Methods for Ship Hydrodynamics", Archives of Computational Methods in Engineering, Volume 18, Number 1, 1-41,

Wang X, Arai M, 2010. "A Study on Coupling Effect between Sea-keeping and Sloshing for Membrane-type LNG Carrier". 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Wang Z, 2010. Personal communication.

Watson S and Bull P, 2008,"Aspects of the Modelling of the Inception of Tip Vortex Cavitation",RINA Symposium on CFD, London, UK

Wemmenhove R, Iwanowski B, Lefranc M, E.P. Veldman A, Luppes R, Bunnik T, 2009. "Simulation of Sloshing Dynamics in a Tank by an Improved Volume-of-Fluid Method". 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Weymouth G, Wilson R, Stern F, 2005. "CFD Predictions of Pitch and Heave Ship Motions in Head Seas". J. Ship. Res.49,80-97

Wilson RV, Carrica PM, Stern F, 2006. "Unsteady RANS Method for Ship Motions with Application to Roll for a Surface Combatant", Comput. Fluids 35, 501-524.

Wilson RV, Nichols DS, Mitchell B, Karman SL, Betro VC, Hyams DG, Sreenivas K, Taylor LK, Briley WR, Whitfield DR, 2007. "Simulation of a Surface Combatant with Dynamic Ship Maneuvers". 9th Int. Conf. Num. Ship Hydrodyn., Ann arbor, MI.

Woeckner K, Drazyk W, Rung T, 2010. "An Efficient VOF-Based RANS Method to Capture Complex Sea States". 29th Intl. Conf. Ocean, Offshore and Arctic Eng.. (OMAE). Shanghai, China.

Wu X, Choi J-K, Hsiao C-T, Chahine GL, 2010, "Bubble Augmented Waterjet Propulsion: Numerical and Experimental Studies ",ONR 2010

Xing T., Carrica P., Stern F., 2008, "Computational Towing Tank Procedures for Single Run Curves of Resistance and Propulsion", Journal of Fluids Engineering, Vol. 130.

Xing, T, Stern, F., 2010, "Factors of Safety for Richardson Extrapolation", J. Fluids Eng, Vol. 132-6, 611403.

Xingrong S, Xuemei F, Rongquan C ,Yuejin C, 2009, "Study on Hydrodynamic Performance of Podded Propulsion in Viscous Flow", SMP09

Yabe, T., Xiao, F. and Utsumi, T. 2001, "The Constrained Interpolation Profile Method for Multiphase Analysis", J. Comput. Phys., 169:556-593

Yamasaki S, Okazaki A, Hasuike N,Kawanami Y, Ukon Y, 2009, "Numerical and Experimental Investigation into Cavitation of Propellers Having Blades Designed by Various Load Distributions near the Blade Tips", SMP09

Yang Q, Qiu W, 2010. "Computation of Slamming Forces on 3-D Bodies with a CIP Method". 29th Intl. Conf. Ocean, Offshore and Arctic Eng.. (OMAE). Shanghai, China.

Zarbock O,2009, "Controllable pitch propellers for future warships and mega yachts", SMP09

Zhang Z.-R., "Verification and validation for RANS simulation of KCS container ship without/with propeller", 2010, Journal of Hydrodynamics, 22(5), pages 932-939.

Zheng X, Maguire J R, Radosavljevic D, 2009. "Validation of Numerical Tools for LNG Sloshing Assessment". 19th Intl. Offshore and Polar Eng. Conf. (ISOPE), Osaka, Japan.

Zorn T, Heimann J, Bertram V, 2010, "CFD Analysis of a Duct's Effectiveness for Model Scale and Full Scale", NutTTs 2010