APPROPRIATE TOOLS FOR FLOW ANALYSES FOR FAST SHIPS

Volker Bertram, Germanischer Lloyd, Hamburg/Germany, volker.bertram@GL-group.com

SUMMARY

An overview of computational tools for the hydrodynamic design of fast ships is given. The individual techniques are discussed, and suitable tools are recommended. Trends are discussed and illustrated by some advanced pioneering applications.

1. INTRODUCTION

In this paper we provide some background and examine some of the developments in fluid dynamics for examining flows in and around ships, where the focus is on fast ships. The work builds on earlier work for general ships, *Bertram and Couser (2007)*. For the scope of this paper we interpret Computational Fluid Dynamics (CFD) to be a numerical, computer-based simulation of a fluid flow, modelled by solving a set of field equations describing the dynamics of the fluid flow. In this context, the field equations are (in increasing order of simplification), *Bertram (2000):*

1. Navier-Stokes equations. For practical problems, the Navier-Stokes equations can only be solved by making certain simplifications leading to the
2. Reynolds averaged Navier-Stokes equations (RANSE). These can be used to solve viscous fluid flows. Removal of the viscous components of the model yields the
3. Euler equations, which are often used in aerodynamic problems where compressibility is important. For ship-flow simulation they are less widely used. Removal of the compressibility terms gives the
4. Laplace and Bernoulli equations (potential flow). Because the effects of viscosity are often limited to a small boundary layer (for streamlined bodies with no separation), potential flow models are very useful, particularly for free surface flows.

Depending on the field equations being solved, different numerical representations of the fluid domain are may be employed. These can be summarised as follows:

1. Field methods – where the whole fluid domain is discretised, namely Finite Element Methods (FEM), Finite Difference Methods (FDM), Finite Volume Methods (FVM)
2. Boundary element methods (BEM)/panel methods – where only the fluid boundary needs to be discretised
3. Spectral methods

While in principle there could be many combinations of field equations and numerical techniques, in practice we see predominantly RANSE solvers based on FVM for solving viscous flows and Laplace/Bernoulli solvers using BEM or simpler analytic-numeric methods for inviscid, potential flow.

Before discussing the tools and trends in more detail, we will briefly discuss the question of whether and when to choose computational approaches and when model tests. Despite all the progress, and despite some marketing claims, computational methods are not able to consistently predict the power requirements of a ship with the same accuracy as model tests performed in professional model basins. CFD offers insight into flow details, overcoming also limitations of scale effects for viscous flows. CFD should thus be used for a preliminary selection of candidate designs and for aiding the design of hull and appendages. The final power prediction for the hull should be based on model tests in professional model basins.

There is a broad range of problems where CFD techniques are applicable; some of the key areas of interest to the naval architect are described below:

- Hull design, especially fore-body design;
- Design of appendages (alignment and form details of shafts, brackets, etc.);
- Propulsor design (efficiency, avoidance of excessive vibrations and cavitation);
- Unsteady ship motions, particularly seakeeping including slamming
- Aerodynamics, HVAC flows, fire simulation

Due to differences in scale, fluid, geometry etc., different CFD techniques are better suited to some problems than others. There is currently no single CFD technique that can be applied to all problems; for this reason, it is generally necessary to have a range of software tools to hand.

While CFD becomes increasingly important for ship design, simpler traditional analysis tools remain popular, since they frequently provide results with sufficient accuracy at low cost. Among these traditional methods are:

- Slender body theories for resistance (only applicable for slender hulls, e.g. catamaran demihulls)
- Strip theory for the prediction of ship motions.
- Coefficient methods for manoeuvring

2. REVIEW OF CURRENT TRENDS IN CFD METHODS

The majority of commercially available CFD codes are either RANSE / FVM for viscous flows and Laplace / BEM for inviscid potential flow. These will be discussed in greater depth in the following.

Potential flow methods are ideally suited to solve the steady 'wave resistance' problem (steady free-surface flow around ship neglecting viscous effects). Computations on a regular PC take typically 10-30 minutes, allowing rapid design exploration. Typically, panels are placed on the submerged part of the ship's hull and the free surface. If the vessel is operating in a confined waterway, the bottom and sides of the channel can also be modelled by including additional panels on these boundaries or by using mirror images of the panels. State-of-the-art fully non-linear wave resistance codes had become standard ship hull design tools by the mid-1990s whilst panel codes for propeller design had reached design maturity even earlier mainly pushed by developments for the aerospace industry.

First-generation wave resistance codes used only source elements to model displacement; propeller codes from the same era used only vortex or dipole elements to model lift. Later developments added lifting surfaces to wave resistance codes (to handle, for example, the keel of a sailing boat) and source elements to propeller codes (to handle thicker blades and the propeller hub). When lifting surfaces are included, it is also necessary to model the trailing vortex wake left downstream. Considerable effort has been made to accurately model the shape and tip roll-up of this wake as this has a significant impact on the accuracy of the induced drag calculation and interaction with downstream bodies.

For many design applications, RANSE solvers with an appropriate semi-empirical turbulence model are sufficient to model a wide variety of ship flows with sufficient accuracy and confidence to be practically useful. The past decade has seen a general trend towards more sophisticated turbulence models, with Reynolds stress models (RSM) and k-ω models now being widely favoured over the older k-ε models. Most RANSE solvers are also able to represent complex free surfaces including breaking waves and air entrapment using volume of fluid (VOF) or, perhaps more commonly, multi-phase flow solutions. RANSE solvers have gained in importance for the analysis of flows around the whole ship hull – an area which until relatively recently used to be the undisputed domain of potential flow solvers – because they can handle the complex geometries of the ship and free surface, including wave breaking.

3. COMMON APPLICATIONS

3.1. Resistance and propulsion (inviscid)

CFD generally gives correct ranking of sufficiently different designs, though absolute values of resistance are normally not accurate enough to exclude the need for towing tank tests. The strength of CFD analysis is that it allows a wider range of alternative hull designs to be tested than would be possible by tank testing alone and is ideally used for selection of promising candidate designs for further testing in the model basin. CFD also gives insight into where and how to modify a design, showing, for example, the detailed pressure distribution over the hull. It is often possible to calibrate a CFD code for a particular design with a "catch all" correlation factor with the experiment results; the correlation factor can be assumed constant for small changes in hull geometry and speed thus allowing further examination of design alternatives using CFD.

The industry workhorse for calculating steady free-surface flows is still the inviscid panel method. The first-generation codes followed Dawson's double-body approach and neither fulfilled the non-linear boundary condition on the free surface nor automatically adjusted the ship to a position of equilibrium. By the end of the 1980's, these drawbacks were overcome with second-generation codes, so-called fully non-linear codes. Amongst the best known of these codes are SHALLO (HSVA), RAPID (MARIN) and Shipflow-XPAN (Flowtech). These codes are regularly used to support design decisions, Fig.1 and Fig.2. They have been successfully applied to a large variety of ship types, including catamarans (with or without foils), frigates, etc.

However, they are not suitable for planing hulls. Over the past decade, these codes have become a standard design tool, increasingly deployed directly at the shipyard by designers rather than dedicated CFD specialists. These codes are particularly useful for the design of the bulbous bow and the forward shoulder of the ship when trying to minimise wave resistance. Although the pressure distribution over the majority of the ship (with the exception of the aft-body) is believed to be quite accurate and wave cuts computed by state-of-the-art codes usually agree well with experiments, the computed wave resistance for real ships may still differ considerably from measured residual resistance or even wave resistance estimated using form factor methods.

Ships with large transom sterns are particularly problematic. There are claims that so-called patch method codes, e.g. KELVIN (SVA Potsdam) and v-SHALLO (HSVA), overcome these shortcomings by providing better resistance prognoses. These codes employ new techniques to improve accuracy, but very little has been published on these codes. However, there seems to be some general improvement in transom stern treatments that allows the typical rooster-tails, found behind fast ships, to be captured. For low to medium speeds, large transom sterns still pose a problem for these inviscid codes. In these situations, a free-surface RANSE simulation is recommended.

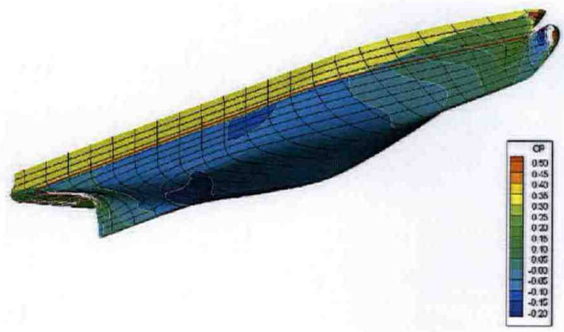

Fig.1: Typical wave resistance code application for fast ferry, HSVA (www.hsva.de)

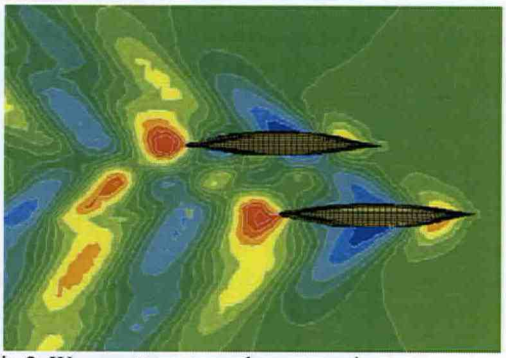

Fig.2: Wave pattern around asymmetric catamaran, HSVA (www.hsva.de)

Usually only the flow fields in the near-field or even in contact with the ship are of interest to the designer aiming to minimize power requirements. However, wave resistance codes have also been used in various projects to develop low-wash ships. This application of panel codes is still in development: design criteria are still to be determined by national and international authorities and the simulations shown so far are usually limited to steady flow conditions neglecting local river topologies and critical unsteady situations such as the deceleration of fast ferries approaching quays. Hybrid methods could be developed matching near-field simulations of the wave generation around the ship (using fully non-linear wave resistance codes or free surface RANSE) and matching the solution to codes used in coastal engineering that simulate the propagation of the wave field in arbitrary shallow-water topology. However, such simulations are rather specific to a particular river or estuary topology. For more general design purposes, a comparison of the near-field wave pattern using a wave resistance cod, usually suffices in practice: if, for a given speed, the waves generated in the vicinity of the ship are reduced, then the wash will also be reduced.

The handling of breaking waves remains a major problem for panel methods, be it for wave resistance or seakeeping. If wave breaking is important, a free-surface RANSE method is the tool of choice. However, maturity, short computational time, ease of grid generation and robustness of the codes explain why panel methods will continue to be the preferred tools for design engineers.

3.2. Resistance and propulsion (viscous)

Flow phenomena such as separation, vortex generation and non-uniformity of the wake field are dominated by viscous effects requiring more sophisticated CFD approaches. In practice, RANSE simulations are normally used where these viscous phenomena are significant. For most design applications, only steady flow is considered.

Most appendages (brackets, rudder, fins, etc.) are located in regions where viscosity cannot be neglected, but where the free surface can be ignored. In these situations CFD allows the simulation at full-scale Reynolds numbers, Fig.3, and thus offers a clear advantage over model tests. The CFD simulation can reveal, for example, how to align propeller shaft brackets so as to minimise resistance and adverse flow patterns in way of the propeller (which cause vibrations).

Similar applications appear for openings in the ship hull such as bow thruster tubes, waterjet inlets etc. Such computations, modelling the flow around appendages, account for a considerable share of viscous flow calculations carried out during design. Although these types of analyses are among the simplest ship applications of RANSE solvers, it is still industry practice to outsource the analysis to experts. This is because the quality of the results is very sensitive to meshing and other analysis parameters which require considerable user experience

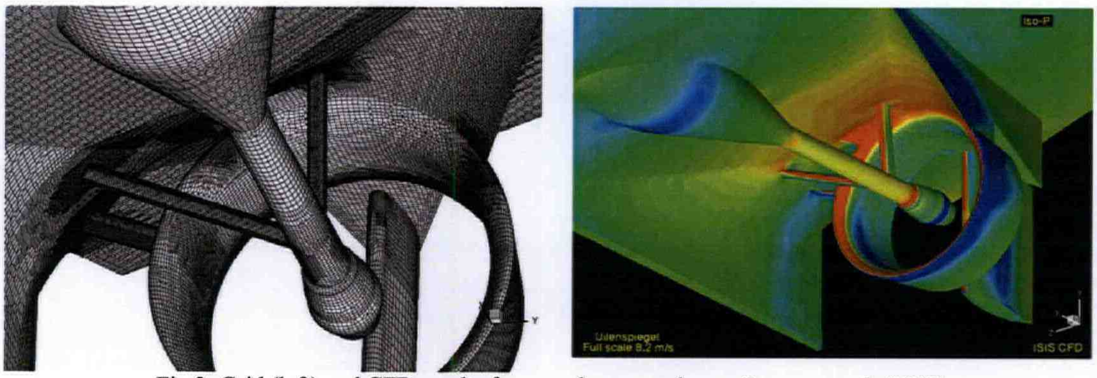

Fig.3: Grid (left) and CFD results for complex appendages, *Queutey et al. (2007)*

RANSE computations that include the effect of propellers (simulated propulsion test) usually model the propeller by applying body forces. Then the propeller geometry is not captured by the grid. Instead each cell in the propeller region is associated with a force representing a contribution to the lateral and rotational acceleration of the water imparted by the propeller. The body forces are often prescribed based on experience or experimental results. Alternatively, panel methods may be employed to predict the thrust and rotation distribution of the propeller. These simulations still appear to be limited to research applications and are not widely used in design. The body force model of the propeller is however frequently employed if the effect of the propeller on appendages in the aft-body is of interest, e.g. for rudders.

The simultaneous consideration of viscosity and wave making has progressed considerably over the past decade. A number of methods try to capture wave making with various degrees of success. The methods for computing flows with a free surface can be classified into two major groups:

- Interface-tracking methods define the free surface as a sharp interface whose motion is followed. They use moving grids fitted to the free surface and compute the flow of the liquid under the free surface only. Problems are encountered when the free surface starts folding or self-intersecting or when the grid has to be moved along walls with complicated shapes (for instance, the geometry of a real ship hull).
- Interface-capturing methods do not define a sharp boundary between liquid and gas and use grids which cover both liquid and gas filled region. The free surface is then determined by either Marker-and-Cell (MAC), Volume-of-Fluid (VOF), level-set or similar schemes.

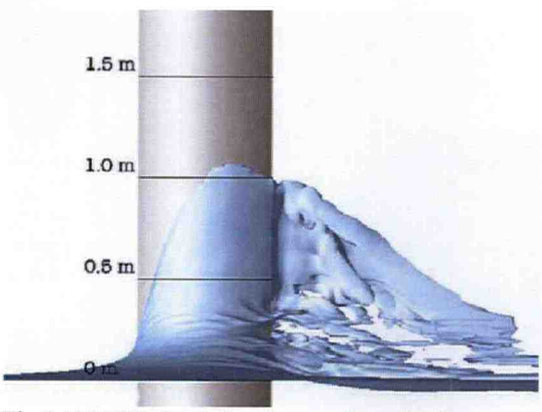

Fig.4: RANSE simulation for a surface-piercing strut, *El Moctar and Bertram (2001)*

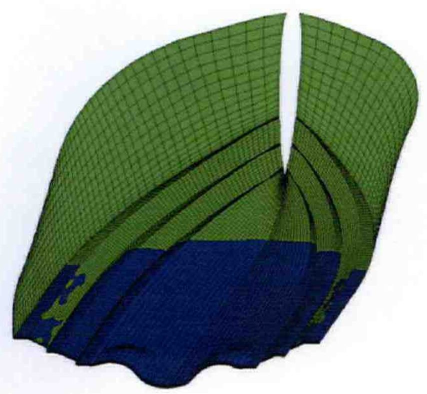

Fig.5: Planing hull simulation, *Caponnetto (2001)*

The trend is clearly towards interface-capturing methods as implemented, for example, in all major commercial RANSE codes. These are the preferred choice whenever wave breaking is of significant importance, e.g. for surface-piercing struts, Fig.4, blunt fore-bodies, etc. Most schemes reproduce the wave profile on the hull accurately, but some problems persist with numerical damping of the propagating ship wave. It is debatable if an accurate prediction of the wave pattern is necessary for practical applications, but certainly everyone would prefer to see this problem overcome. This may require considerably finer resolution and higher-order differencing, i.e. much higher computational times and storage capacities. For global wave system creation, the much cheaper wave resistance codes seem sufficiently accurate and are our recommended tool of choice.

For planing hulls, the classical Savitsky approach remains popular. However, real planing hull geometries violate the inherent assumptions of Savitsky's approach, e.g. concerning constant deadrise angle over the length of the hull. Free-surface RANSE computations yield good results, Fig.5, e.g. *Caponnetto (2000,2001)*. However, such computations require considerable skill (experience with the code), hardware (parallel clusters) and expensive software. The average designer is left with the choice between outsourcing these analyses to a few specialists worldwide or to live with significant errors in traditional simple methods.

3.3. Propeller

Inviscid flow methods (panel methods and vortex lattice methods) have long been used in propeller design as a standard tool yielding information comparable to experiments. Today, RANSE methods also yield good results for 'nice' propeller geometries. However, both panel methods and RANSE deteriorate for extreme propeller geometries due to grid problems. Also, certain types of cavitation still are not satisfactorily reproduced by the computations. Free-surface RANSE method are able to simulate also surface-piercing propellers, Fig.6, *Caponnetto (2003)*. Special propulsors such as waterjets are best analysed using RANSE methods, Fig.7.

Most publications for propeller flows focus on open-water simulations. In practice, the propeller should be designed for the effective wake field of the full-scale ship, considering hull-propeller and propeller-rudder interactions. Complete RANSE simulations appear to be unnecessarily expensive and so far yield results no better than hybrid approaches that combine potential flow computations and RANSE.

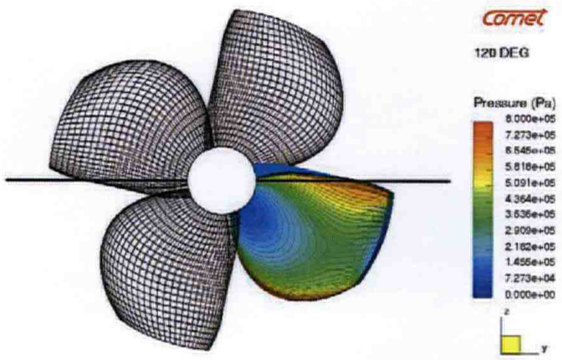

Fig.6: Surface-piercing propeller, *Caponnetto (2003)*

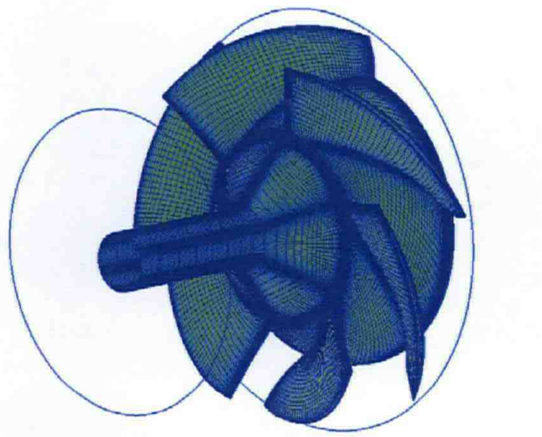

Fig.7: Grid for impeller in waterjet, *Seil (2003)*

3.4. Seakeeping

Although the underlying physical models are generally considered crude, strip methods are able to calculate most seakeeping properties of practical relevance accurately enough for displacement monohulls. Strip methods are generally applicable up to Froude numbers of 0.4. With some corrections, this range can be extended up to Froude numbers of 0.6. For displacement hulls at Froude numbers above 0.4, 2D+t methods (also called high-speed strip methods HSST) are fast and yield good results, *Bertram and Iwashita (1996)*. *Söding (1988,1999)* developed a strip method for catamarans named SEDOS, Fig.8. However, the software is not available and the theory apparently too complex to reproduce.

For catamaran seakeeping, no simple recommendation can be given. 3-d potential flow codes for seakeeping are usually based on Green function methods (GFM). These work well for zero and low Froude numbers, but are computationally expensive for high Froude numbers, unless (unphysical) simplifications are introduced. These code frequently also neglect the real average floating position of the vessel at design speed and compute for the zero-speed floating position. For comparative evaluations, for heave and pitch motions, this approach is OK. Alternatively, 3-d Rankine Singularity Methods (RSM) may be used, but these have problems to enforce correct wave propagation for all speed-frequency combinations in frequency domain and are time-consuming in time-domain simulations.

Some pioneering applications of RANSE computations for ships in regular waves have appeared. Computing power is now the main limiting factor: even when powerful computer clusters are employed, simulations are limited to a few seconds. RANSE simulations make sense for strongly non-linear cases involving green water on deck and slamming, Fig.9, *Fach and Bertram (2006).*

Seakeeping of planing hulls is one area where RANSE simulations would be our recommended choice. Rolla Research in Switzerland and MTG in Germany have presented convincing applications for real planing hull geometries, *Caponnetto (2001), Caponnetto et al. (2003).* The RANSE code employed (COMET in both cases) was reported, in personal communication, to give "good results in 9 out of 10 cases", but such an analysis requires considerable experience with RANSE codes and significant hardware resources, forcing designers to outsource the services to select experts.

Slamming problems, even in two dimensions are very challenging. They involve rapidly changing local hull loads; hydro-elastic effects; interaction between trapped air pockets and the surrounding water; compressibility of water in localised regions, leading to the formation of shock waves; and complex water surface shapes due to the formation of jets. Traditional approaches work well for two-dimensional flows around wedges of suitable deadrise angle, but real ships are 3-d and do not have 'suitable' deadrise angles! CFD simulations have progressed immensely over the last decade, but are still limited to research applications. None of the methods developed so far incorporate all relevant phenomena and adaptive grid techniques appear mandatory to allow realistic computations in an acceptable time. Designers will continue to use the recommendations made by classification societies, which are in turn developed using a mix of full-scale experience, model tests and advanced simulations.

In practice, the ship designer will probably use strip methods for most problems. RANSE methods or non-linear strip methods may be employed by experts for a few specific, highly non-linear problems.

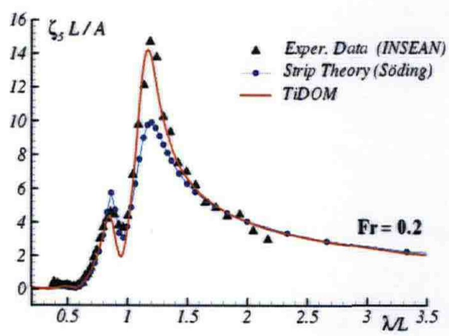

Fig.8: 3-d RSM and multihull strip method applied to a trimaran, *Landrini and Bertram (2002)*

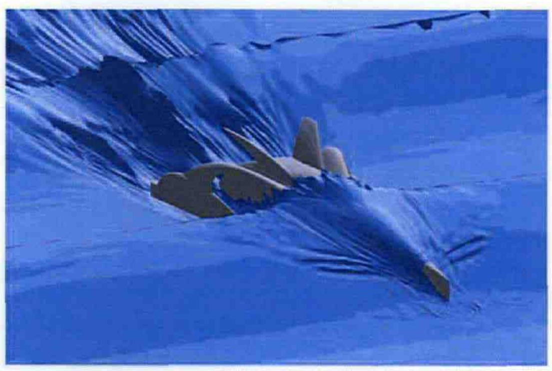

Fig.9: 'Earthrace' trimaran piercing through waves in RANSE simulation, *Ziegler et al. (2006)*

3.5. Manoeuvring

CFD simulations of ship manoeuvring remain limited to advanced research applications. For practical applications, the preferred choice is a force-coefficient method that employs various coefficients to approximate the forces acting on the ship (hull, rudder, propeller, thrusters, etc), *Bertram (2000).* Some of these coefficients can be predicted accurately by CFD, but usually empirical estimates or computations based on slender-body theory suffice.

However, CFD has gained rapid acceptance for rudder design. For many applications, potential flow models enhanced by empirical corrections are sufficient, but for large rudder angles (where the onset of separation is approached) and partially cavitating flows, RANSE simulation is the tool of choice, Fig.10. The designer strives to avoid rudder cavitation for rudder angles up to ±5°. This is the usual operating range for rudders during normal ship course keeping. Cavitation is almost unavoidable for highly loaded rudders at large rudder angles and in these situations it is normal practice to accept it. Modern RANSE codes with cavitation models predict location and extent of cavitation on rudders at full scale very well, Fig.11, *GL (2005), El Moctar (2007).*

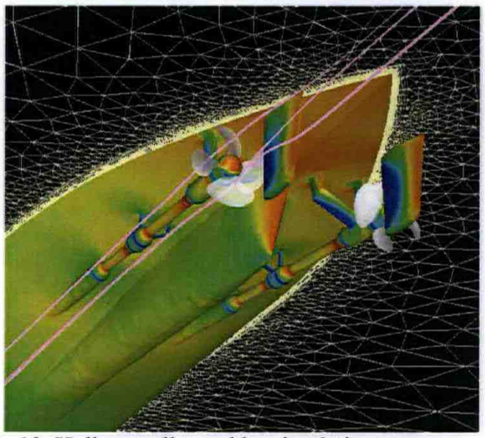

Fig.10: Hull-propeller-rudder simulation, *Hino (2007)*

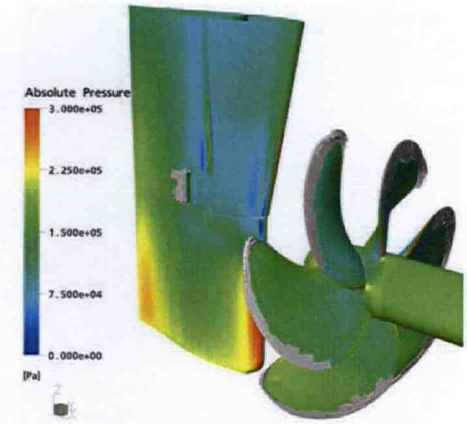

Fig.11: Cavitation on rudder, *El Moctar (2007)*

3.6. Aerodynamics, HVAC and Fire simulations

CFD may be applied to the airflow around the upper hull and superstructure of ships. Topics of interest are wind resistance, wind-over-the-deck conditions for helicopter landing, wind loads and tracing of funnel smoke. The differences between CFD and model-test results are not generally larger than between full-scale and model-scale results. However, due to the time involved in generating the computational mesh and in computing the flow patterns, CFD is usually not economically competitive when compared with routine wind tunnel model tests. For wind forces, empirical estimates usually work well enough for most ships. With decreasing time and cost of grid generation around complex ship super-structures, we may see more CFD applications for ship aerodynamics, but so far such simulations are only applied in research or in combination with other features, for example fire and ventilation flow simulations. Our tool of choice remains thus a wind tunnel in most design applications.

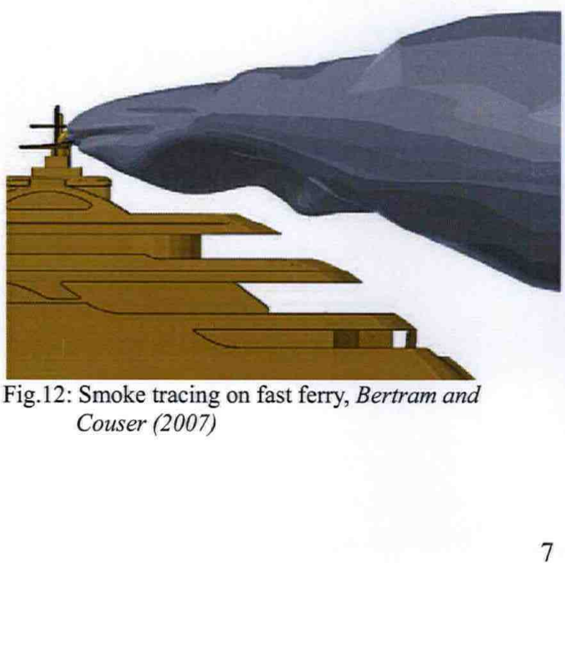

Fig.12: Smoke tracing on fast ferry, *Bertram and Couser (2007)*

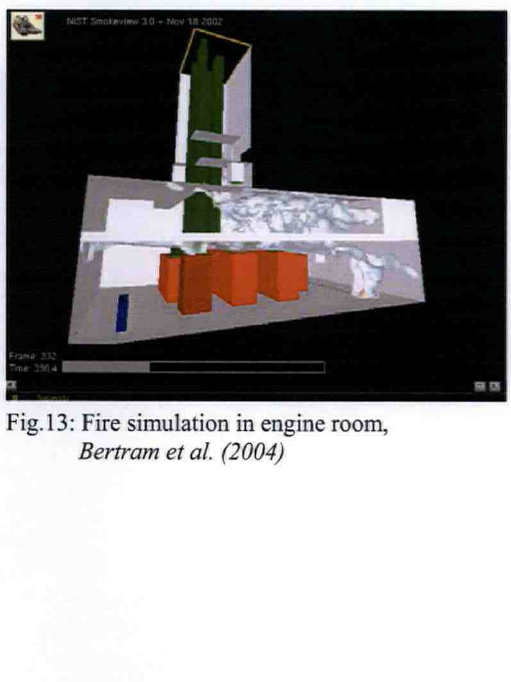

Fig.13: Fire simulation in engine room, *Bertram et al. (2004)*

For fire simulations in ships, different tools are employed, solving additional equations that describe the energy aspects and the combustion (chemical reaction). Applications have graduated from preliminary validation studies to more complex applications for typical ship rooms, e.g. *Bertram et al. (2004).*

The simulations are able to reproduce qualitatively all major fire characteristics, but presently available software and hardware do not yet yield reliable quantitative predictions, particularly not for larger and complex geometries. However, a lot more progress can be envisioned in the next decade and the fire simulations appear already suitable to give some general support both for fire containment strategies and for design alternatives.

The experience of hydrodynamic or aerodynamic flows is not directly transferable to fire simulations. Therefore, fire simulations should be left to experts, preferably those with experience in modelling such scenarios onboard ships.

4. IN-HOUSE OR OUTSOURCE?

Many of the software vendors provide consulting services, and there are specialist consultants and model basins which will perform CFD analyses. The quality of the results depends generally more on the skill of the operator than on the CFD tool used. Sufficient experience with the software, particularly the grid generation, is the decisive factor for the cost and quality of the analysis. As a simple rule of thumb: it becomes cost-effective to do the analyses in-house if you perform more than ten analyses per year and you are able to stay sufficiently up-to-date with the software and technology. If you only perform CFD analyses infrequently, it is advisable to outsource the analysis when the need arises.

To be able to use advanced CFD applications in-house requires:

- Specialist CFD staff, typically requiring several months training to become proficient in the use of an analysis package.
- Software licences for grid generators, flow solvers and post-processing tools (and possibly further codes);
- Significant computer resources, typically distributed PC clusters

This type of investment only pays off if CFD analyses are performed on a regular basis. Vendors frequently downplay the cost of initial training. For design offices and independent shipyards, there is little sense in using RANSE codes; it will normally be more cost-effective to outsource these analyses to specialists. However, inviscid, potential flow, wave resistance codes can be recommended for in-house use if there are ten or more projects per year. Similarly strip methods (or high-speed strip method for fast ships) for seakeeping analyses make sense because the codes can be run on standard PCs, generation of the input data is fast and relatively simple. In any case, generation of input data and interpretation of the result requires an understanding of the fundamental theory behind the code and its assumptions and limitations.

If you decide to buy software and use it in-house, we recommend using commercial software with large user groups in the shipbuilding industry. Commercial codes have the advantage of large user community pools of experience. This usually reduces the (re)occurrence of mistakes. This is not a general law, but a frequently observed fact. Also commercial codes are usually better validated and documented. The larger user community supports continuous development and enhancement of the software, in terms of both features and ease of use. From a business point of view, commercial codes often make more sense than one-off products fresh from universities or in-house researchers.

In evaluating different software products, pay attention to grid generation tools used. Grid generation is usually the most time-consuming (and thus expensive) part of each CFD analysis. Additional licences may be necessary for appropriate professional grid generators. Integrated CFD environments are the most user-friendly option. A noteworthy example is FRIENDSHIP-Framework, *Abt and Harries (2007).*

5. REFERENCES

ABT, C.; HARRIES, S. (2007), *A new approach to integration of CAD and CFD for naval architects*, 6th Conf. Computer and IT Applications in the Maritime Industries (COMPIT), Cortona, pp.467-479, http://www.compit07.insean.it/proceedings/Proceedings.pdf

AZCUETA, R. (2003), *Steady and unsteady RANSE simulations for planing crafts*, 7th Conf. Fast Sea Transportation (FAST), Ischia, http://azcueta.net/Publications/Azcueta_FAST03.pdf

BERTRAM, V. (2000), *Practical ship hydrodynamics,* Butterworth-Heinemann, Oxford, ISBN-13: 978-0750648516

BERTRAM, V.; COUSER, P. (2007), *CFD possibilities and practice*, The Naval Architect, September, pp.137-147

BERTRAM, V.; EL MOCTAR, O.M.; JUNALIK, B.; NUSSER, S. (2004), *Fire and ventilation simulations for ship compartments,* 4th Int. Conf. High-Performance Marine Vehicles (HIPER), Rome, pp.5-17

BERTRAM, V.; IWASHITA, H. (1996), *Comparative evaluation of various methods to predict seakeeping qualities of fast ships,* Schiff+Hafen 48/6, pp.54-58

CAPONNETTO, M. (2000), *Numerical simulation of planing hulls*, 3rd Num. Towing Tank Symp. (NuTTS), Tjärnö, http://www.rolla-propellers.ch/downloads/nutts_2000.pdf

CAPONNETTO, M. (2001), *Practical CFD simulations for planing hulls*, 2nd Conf. High-Perfor-mance Marine Vehicles (HIPER), Hamburg, pp.128-138, http://www.rolla-propellers.ch/downloads/hiper_2001.pdf

CAPONETTO, M. (2003), *RANSE simulations of surface piercing propellers,* 6th Numerical Towing Tank Symposium (NuTTS), Rome, http://www.rolla-propellers.ch/downloads/nutts_2003.pdf

CAPONNETTO, M.; SÖDING, H.; AZCUETA, R. (2003), *Motion simulations for planing boats in waves,* Ship Technology Research 50, pp.182-198

EL MOCTAR, O.M. (2007), *How to avoid or minimize rudder cavitation*, 10th Num. Towing Tank Symp. (NuTTS), Hamburg

EL MOCTAR, O.M..; BERTRAM, V. (2001), *RANSE simulations for high-Fn, high-Rn free-surface flows*, 4th Numerical Towing Tank Symposium (NuTTS), Hamburg

FACH, K.; BERTRAM, V. (2006), *High-performance simulations for high-performance ships*, 5th Int. Conf. High-Performance Marine Vehicles (HIPER), Launceston, 2006, pp.455-465

GL (2005), *Recommendations for preventive measures to avoid or minimize rudder cavitation*, Germanischer Lloyd, Hamburg,

HINO, T. (2007, *Marine CFD research at SRI/NMRI – Review and prospects,* 10th Numerical Towing Tank Symposium (NuTTS), Hamburg

LANDRINI, M.; BERTRAM, V. (2002), *Three-dimensional simulation of ship seakeeping in time domain*, Jahrbuch der Schiffbautechnischen Gesellschaft, Springer

QUEUTEY, P.; DENG, G.; VISONNEAU, M. (2007), *Study of scale effects around fully-appended ships with an unstructured RANSE solver*, 6th Conf. Computer and IT Applications in the Maritime Industries (COMPIT), Cortona, pp.260-274

SEIL, G.J. (2003), *RANS CFD for marine propulsors: A Rolls-Royce perspective*, 6th Numerical Towing Tank Symposium (NuTTS), Rome

SÖDING, H. (1988), *Berechnung der Bewegungen und Belastungen von SWATH-Schiffen und Katamaranen im Seegang*, IFS Report 483, Universitat Hamburg (in German)

SÖDING, H. (1999), *Seakeeping of multihulls*, 1. Int. Conf. High-Performance Marine Vehicles (HIPER), Zevenwacht

ZIEGLER, W.; FACH, K.; HOFFMEISTER, H., EL MOCTAR, O.M. (2006), *Advanced analyses for the EARTHRACE project*, 5th Conf. High-Performance Marine Vehicles (HIPER), Launceston, pp.101-108