UDC 532.5:629.51/.58

Rojko DEJHALLA
Jasna PRPIĆ-ORŠIĆ

A Review of the State-of-the-Art in Marine Hydrodynamics

Review

The purpose of the present paper is to summarise the present situation in the field of marine hydrodynamics. Since the William Froude time there has been considerable development in all fields of marine hydrodynamics, both experimental and particularly in theoretical methods and their numerical implementation. The role of computational methods is becoming more important because modern technology requires analysis of increasingly complex phenomena. The hydrodynamics institutes make efforts to expand their activities through integration of experimental and computational approach in order to be able to successfully answer the increased demands of the related industries.

Key words: *marine hydrodynamics, computational fluid dynamics, state-of-the-art.*

Pregled stanja u području hidrodinamike pomorskih objekata

Pregledni rad

Svrha rada je dati pregled postojećih dostignuća u području hidrodinamike pomorskih objekata. Od vremena Williama Frouda postignut je značajan razvitak u svim područjima hidrodinamike, u eksperimentalnim metodama te posebice u teorijskim metodama i njihovoj numeričkoj primjeni. Uloga računarskih metoda postaje sve važnija budući da moderna tehnologija zahtijeva analizu sve složenijih pojava. Hidrodinamički instituti ulažu napore u proširenje svojih aktivnosti kroz integraciju eksperimentalnog i računarskog pristupa kako bi bili u mogućnosti uspješno odgovoriti na povećane zahtjeve zainteresiranih industrija.

Ključne riječi: *hidrodinamika pomorskih objekata, pregled stanja, računarska dinamika fluida*

Authors' address:
Faculty of Engineering
University of Rijeka
Department of Naval Architecture and Ocean Engineering
Vukovarska 58
HR-51000 Rijeka, Croatia
E-mail: roko@riteh.hr;
jasnapo@riteh.hr

Received (Primljeno): 2005-11-03
Accepted (Prihvaćeno): 2006-01-09

1 Introduction

In the course of the 19th century arose a strong need for a method of determining beforehand the power to be installed in a certain ship to reach a certain speed and this problem had already engaged the attention of many scientists of note. It was natural that in searching after a solution the thoughts of several among them turned to making model experiments. At that time, however, the correct physical insight into the flow phenomena around a moving ship was still lacking. Consequently, the only result obtained by all these experimenters was that the results of model experiments were discredited by the navies and shipping companies to such an extent that every penny spent on them was considered as having been thrown away. This situation was ended at a single stroke by the genius of an English naval architect William Froude (1810-1879). This founder of the entire experimental technique with regard to the examination of hull forms and propellers was the first to gain from experiments a fundamental insight into the character of the phenomena occurring. He succeeded, thought to a limited extent, in obtaining the support of the British Admiralty, and thus the first model experiment tank in the world was built at Torquay about 1870. It was Froude's ingenious understanding, which enabled model experiments to be carried out in a manner, which was physically justified, and the behaviour of the actual ship to be derived from these experiments. It is significant that in principle his method is still in use

Pregled stanja u području hidrodinamike pomorskih objekata

Od vremena W. Frouda prisutan je snažan razvoj u različitim područjima hidrodinamike pomorskih objekata. Umjesto promatranja ukupnoga fenomena strujanja pozornost se usmjerava prema iscrpnom poznavanju fizike problema, a nove eksperimentalne tehnike pružaju nove mogućnosti ispitivanja modela. Premda je od vremena Frouda bilo malo pomaka u osnovnoj metodologiji određivanja otpora broda, znatno su se razvile brojne metode i postupci. Nastavljeni su izgradnja novih i obnova postojećih eksperimentalnih objekata, razvoj novih postupaka za izradu modela, kao i novih mjernih tehnika. Mnogi bazeni za ispitivanje modela obnovom su dobili mogućnost ispitivanja pri velikim brzinama. Za izgradnju novih ili obnovu postojećih bazena kapitalna financijska sredstva većinom osiguravaju razne državne agencije. Eksperimentalni pristup prolazi promjene od rutinskih ispitivanja za određivanje globalnih veličina k iscrpnom mjerenju lokalnih veličina što se koristi za razvoj modela. Time se metodologija projektiranja mijenja od ispitivanja modela i teorijskoga pristupa ka projektiranju na temelju simulacija.

Ovakva iscrpna ispitivanja zahtijevaju da bazeni za ispitivanje modela koriste naprednu suvremenu opremu s dokumentiranim uvjetima ispitivanja, postupcima i procjenama pouzdanosti. Modelska ispitivanja koja se obavljaju u hidrodinamičkim laboratorijima znatno su napredovala i danas se mogu razvrstati na sljedeći način:

- Fundamentalna ispitivanja radi razjašnjavanja temeljnih pojava;
- Korelacijska ispitivanja namijenjena za ocjenu valjanosti računarskih metoda;
- Sustavna ispitivanja čija je svrha prikupljanje hidrodinamičkih podataka za sustavno mijenjanje parametara radi dobivanja iskustvenih koeficijenata primjerice za ugrađivanje u računarske programe za predviđanje značajki;
- Ispitivanja izvodljivosti radi potvrđivanja procjene koncepcije novoga broda prije daljeg razvoja koncepcije;
- Projektna ispitivanja, s ciljem dobivanja podataka koji se posebno tiču određenoga projekta.

U prošlosti su u hidrodinamičkim laboratorijima najčešće bila zastupljena projektna ispitivanja dok danas sve veća važnost numeričkih proračuna uvjetuje povećanje broja korelacijskih ispitivanja.

Otkad je po prvi put prije nekih 20 godina dobila na važnosti, računarska dinamika fluida (RDF) postala je moćni inženjerski i projektantski alat u automobilskoj i zrakoplovnoj industriji. Dostupnost snažnih komercijalnih računarskih programa RDF kao i brzih računala dovela je do povećane primjene RDF u cilju traženja rješenja za inženjerske probleme u svim industrijskim granama, pa tako i u industriji pomorskih objekata. RDF zauzima mjesto između analitičke i eksperimentalne dinamike fluida (EDF). Od njih se razlikuje, premda imaju zajednička polazišta, te ih prije nadopunjuje nego što im konkurira. RDF omogućuje izvođenje numeričkoga pokusa na način vrlo sličan fizičkom pokusu, kao na primjer u bazenu za tegljenje modela. RDF ne daje funkcijsku ovisnost kao što to omogućuje dobra teorija i u tom smislu ima slične nedostatke kao fizički pokus. RDF se u međuvremenu razvila u samostalnu disciplinu unutar dinamike fluida, različitu i od teorije i od eksperimenta, s vlastitim metodama, vlastitim teškoćama, ali također i s vlastitim probitkom. Kod primjene RDF na pomorske objekte od glavne su važnosti primjene na hidrodinamičke probleme. U većini problema koji se rješavaju, nastoji se izračunati globalne tlakove i sastavnice brzine fluida u trodimenzijskom prostoru koji okružuje uronjeni dio plovnog ili nekog drugog pomorskog objekta koji se razmatra. Na taj je način moguće dalje računati sile i momente koji djeluju na plovni objekt ili pomorsku konstrukciju. Prisutnost slobodne površine kod primjene u hidrodinamici pomorskih objekata predstavlja glavnu razliku od primjene RDF na konvencionalne probleme, a potreba za preciznim uključivanjem ove granice fluida predstavlja vrlo velik izazov. Primjena računarskih metoda u hidrodinamici pomorskih objekata obuhvaća proračune viskoznoga kao i neviskoznoga strujanja. Općenito se primjenjuju metode pogodne za proračun nestlačivoga, stacionarnog i nestacionarnog, laminarnog i turbulentnog strujanja, sa i bez uključene slobodne površine. Raspon alata RDF koji su na raspolaganju za ovu vrstu problema širok je i raznolik, a njihov je razvoj slijedio različite putove. Upotrebljavaju se programski paketi RDF specijalizirani za pomorske

in all model experiments tanks. By the end of the 19th century tanks were established in Great Britain, Germany, Italy, the Netherlands, Russia and the United States of America. Simple wave makers were added to one end, to enable tests to be conducted in regular waves. Attention was also directed to tests on propellers, and circulating water tunnels were developed. The increasing interest in systematic experiments at a number of different facilities led naturally to discussions about consistency and in 1933 tank superintendents from a number of countries met for the first time under the auspices of what was to be later called the International Towing Tank Conference (ITTC). This organisation has provided the international focus for efforts to standardise on such matters as tank testing methodology and presentation of results.

When it first became prominent some 20 years ago, computational fluid dynamics (CFD) became a powerful engineering and design tool in the hands of the automotive and aerospace industries. It remains a force in both those fields and among the top 20 automotive companies CFD consumes about 25 percent of all supercomputing cycles. In airframe design, CFD is a bedrock technology for studies of aerodynamics, cabin ventilation, combustion, hydraulics and other areas. The availability of robust commercial CFD software and high speed computing has lead to the increasing use of CFD for the solution of fluid engineering problems across all industrial sectors and the marine industry is no exception. CFD holds a position between analytical and experimental fluid dynamics (EFD). It is distinct from each although it has aspects in common with both, and it supplements them rather than being competitive. CFD makes it possible to carry out a numerical experiment in a manner very similar to physical experimentation, for instance, in towing tank. CFD does not provide functional relationships as does a good theory, and in this sense it has a similar drawback as a physical experiment. In short, CFD has evolved in the discipline *per se* in fluid dynamics, distinct from theory and experiment, with its own techniques, its own difficulties, but also with its own realm of utility.

Much can now be achieved in simulating the flows past marine vehicles and other marine structures, and using such simulations to predict the implications of modifying a design in some controlled manner. The extent to which these scientific and technological developments are being used in design is rather variable, depending on the marine application. Their adoption by the designers of conventional merchant vessels has been disappointingly slow. Ship construction has a long history of evolutionary design, over many centuries during which scientific methods were not available. Younger industries, such as aviation or offshore industry, have been able to benefit rapidly from the application of science to technological development. These industries had no historical database to rely on, and were forced to adopt a scientific approach. Rapid scientific and technological progress is nevertheless being made, driven in large measure by the defence and offshore industries, and the recent interest in non-conventional merchant vessels, such as high-speed catamarans. The paper describes some of the progress in the field of marine hydrodynamics, paying attention to the use both of physical models and particularly of the advanced tools of CFD.

2 Experimental approach

Since the time of W. Froude there has been considerable development in various fields of marine hydrodynamics. Attention shifts from overall flow phenomena towards detailed physics and new experimental techniques provide new possibilities in model testing. In addition to the development of modern towing tanks, from the 1950s onwards special purpose tanks have been designed to investigate manoeuvring, propeller performance, and seakeeping of a variety of types of marine vehicles. Initially straight line tests in a towing tank were used to determine forces and moments on a captive model of a ship and rudder at an angle of attack to the incident flow; and hence to predict her trajectory during manoeuvring. To measure corresponding forces and moments associated with a yaw angular velocity, wide tanks have been established, in which a rotating arm experiment can be conducted. More recently, with advances in control systems and instrumentation, manoeuvring experiments have also been conducted using free running models in large enclosed basins. In the interests of improving propeller performance, tests are now often conducted in cavitation tunnels. Experiments on seakeeping are now very often conducted in pseudo-random waves. These are generated to match a prescribed wave spectrum, by means of digitally controlled actuators driving a wedge, flap or plunger. With the increasing interest since the 1970s in the behaviour of offshore facilities fixed at one location, it has become necessary to investigate the effects of directional spreading in the ambient wave conditions. Very long structures designed as platforms for industrial plant are also sensitive to the effects of wave spreading. Therefore, large wave basins which allow simulation of a target wave spectrum as a function of both frequency and direction have been developed.

Today many model test facilities exist around the world, almost all of them are fitted with a ship model towing tank. It is not possible here to include an exhaustive list of these facilities, but some of the main features of a range of towing test facilities are summarised in Figures 1 and 2.

The length of most towing tanks is in the range 70-250 m which accommodates models with a length between 2 and 6 m running at a maximum speed of 2-16 m/s. Smaller tanks are usually associated with educational and research establishments and the larger ones with marine research and development centres. More than 50% of towing tanks have a wave maker, mostly

objekte, kao i inženjerski alati RDF namijenjeni za općenitije namjene. Potreba za razumijevanjem fizike problema koji se razmatra, ograničenja jednadžbi koje se koriste, osnova primijenjenih numeričkih metoda i postupaka u cilju dobivanja što točnijih i dosljednijih rezultata samo su neki od zajedničkih izazova s kojima se korisnici RDF u području pomorskih objekata kao i u drugim inženjerskim područjima susreću. Brodograđevna i *offshore* industrija pokazuju široko zanimanje za RDF i prepoznaju njezinu svestranost za rješavanje praktičnih inženjerskih problema. Pritom predlažu sljedeće općenite teme koje bi trebalo razmotriti u budućim istraživanjima:

- Integracija alata i postupaka za sve sadržaje u procesu analize (potencijalno strujanje i vremenski po Reynoldsu osrednjene Navier-Stokesove jednadžbe, kruto tijelo i strukturna analiza);
- Poboljšanje brzine i točnosti rješavača;
- Ocjena valjanosti modela RDF usporedbom s ponašanjem objekata u pravoj veličini primjenom usavršenih postupaka mjerenja i opreme;
- Uvođenje optimizacijskih alata u cilju optimizacije oblika i značajki.

U hidrodinamici pomorskih objekata prisutan je brzi napredak zbog novog razvoja koncepcija brodova i pomorskih konstrukcija, te projektnih alata, metoda RDF i njihova korištenja za projektiranje, te sofisticiranih mjernih sustava EDF i njihova korištenja u složenijim modelskim ispitivanjima lokalnoga strujanja. Hidrodinamički instituti znatno proširuju svoje aktivnosti izvan rutinskih ispitivanja, te koriste prikupljeno stručno znanje za nadopunjavanje RDF i EDF. Nakon stotinu godina oslanjanja na eksperimente kod projektiranja brodova i pomorskih konstrukcija, industrija ulazi u razdoblje kada će numerički modeli moći ponuditi vjerodostojnu i učinkovitiju alternativu za širok raspon hidrodinamičkih problema kod projektiranja pomorskih objekata. To međutim ne uklanja potrebu za ispitivanjem modela i prikupljanjem podataka za objekte u pravoj veličini. Jedino uz spregu numeričkih proračuna i eksperimenata moguće je ocjenjivati valjanost računarskih modela. Da bi to postalo djelotvorno, mora postojati sinergija između eksperimentatora i numeričara.

Figure 1 **Towing tank length**
Slika 1 **Dužine bazena za tegljenje modela brodova**

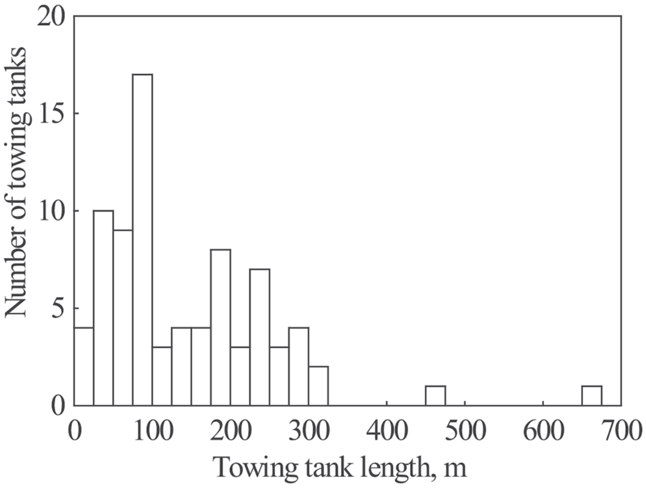
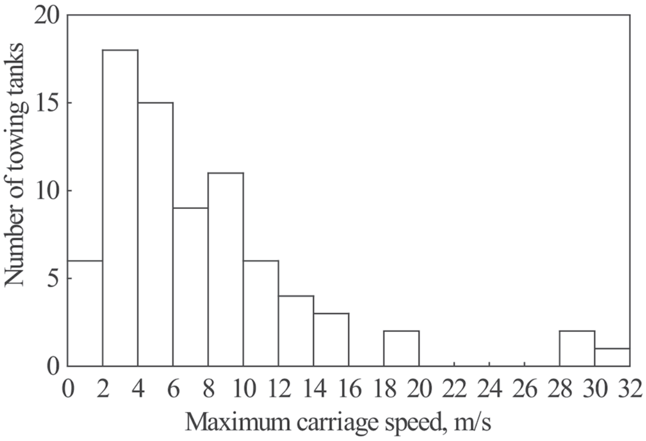


Figure 2 **Carriage speed**
Slika 2 **Brzina kolica**



capable to create irregular waves up to the significant wave height from 0.1 m to 1.0 m. About 10% of towing tanks are equipped with wind blowers.

The basic idea of model testing is to experiment with a scale model to extract information that can be scaled to the full-scale ship. Despite continuing research and standardisation efforts, a certain degree of empiricism is still necessary, particularly in the model-to-ship correlation which is a method to enhance the prediction accuracy by empirical means. Traditionally, model basins tend to adopt approaches that seem most appropriate to their respective experience and accumulated databases. Unfortunately, this makes various approaches and related aggregated empirical data incompatible.

Although there has been little change in the basic methodology of ship resistance since the days of Froude, various aspects of the techniques have progressed a lot. There is now better understanding of the flow around appended ship, and especially of boundary layer effects. Also non-intrusive experimental techniques like laser-Doppler velocimetry (LDV) allow the measurement of the velocity field in the ship wake to improve propeller design. Another experimental technique is wave pattern analysis to determine the wave-making resistance. In propulsion tests, measurements include towing speed and propeller quantities such as thrust, torque, and rpm. Normally, open-water tests on the propeller alone are run to aid the analysis process as certain coefficients are necessary for the propeller design. The powering performance of a ship is validated by actual ship trials, ideally conducted in calm seas. Since the 1990s, the global positioning system (GPS) and computer-based data acquisition systems have considerably increased the accuracy and economy of full-scale trials. The GPS has eliminated the need for "measured miles" trials near the shore with the possible contamination of data due to shallow-water effects. Today trials are usually conducted far away from the shore. Model tests for seakeeping are often used only for validation purposes. However, for open-top containerships and RO-RO ships model tests are often performed as part of the regular design process, as IMO regulations require certain investigations for ship safety, which may be documented using model tests. Most large model test facilities have a manoeuvring model basin. The favoured method to determine the coefficients for the equations of motion is through a planar motion mechanism and rotating arm model tests. Also, manoeuvring tests have been carried out with radio-controlled models in lakes and large reservoirs. These tests introduce additional scale effects, since the model propeller operates in a different self-propulsion point than the full-scale ship propeller. In general, ship manoeuvrability at design stage is estimated using theoretical methods, model experiments or both, but there is no guarantee that the ship will behave identically at sea, because of errors and uncertainties such as scaling and environmental disturbances. Consequently, the predicted manoeuvre should be validated by comparison with full-scale trials. As the study of ship behaviour in confined areas often requires model tests, experimental facilities were developed in the early nineties. At present they consist of a shallow-water towing tank, equipped with a planar motion carriage, a wave generator and an auxiliary carriage for ship-ship interaction tests.

New construction and renovation of existing experimental facilities, the development of new manufacturing systems for models and of new measurement techniques continue. In renovation of existing towing tanks, most of them have been aimed at high-speed testing. In today's world of Internet, computers on a towing carriage are connected to the local area network (LAN) on land, and the wireless LAN is the most common communication tool between the towing carriage and land. For most of new constructions or renovations the capital funding has been provided largely by government agencies.

Some of the towing tanks newly built or renewed in the past decade are:

- In 1995 a new towing tank (length × width × depth = 235.4 m × 11 m × 5.5 m) at *IHL, BPPT*, Indonesia;
- In 1996 a new towing tank (length × width × depth = 400 m × 14 m × 7 m) at *SSMB*, Korea;
- In 2001 a new towing tank (length × width × depth 545 m × 15 m × 7 m) at *Bassin d'Essais des Carenes*, France;
- In 2001 renewal of a towing tank (length × width × depth = 148 m × 5 m × 3 m) at *Ecole Centrale de Nantes*, France;
- In 2001 renewal of a depressurised towing tank at *MARIN*, the Netherlands;
- In 2001 a new towing carriage in the shallow water basin (rail length = 150 m) at *VBD*, University of Duisburg, Germany;
- In 2005 renewal of a towing tank (length × width × depth = 100 m × 3.55 m × 1.5 m) at *AMC*, Australia.

New experimental techniques in ocean engineering are strongly demanded in the field of deep-sea technology. Sub-sea oil fields in the shallow water, easy accessible large reservoirs are getting fewer, and new discoveries are made in deeper and deeper waters. Researches on the dynamics and control of umbilicals, pipelines and riser systems are very important for developments of deep-sea oil fields. For these researches, deep-water experimental tanks have been constructed and as an example the new deep-water tank (1999) at MARIN, the Netherlands can be mentioned here. The dimensions of the tank are 45 m × 36 m × 10.5 m with an additional deep pit (30 m), in which wind, waves and current are generated to simulate the realistic environment. In 2001 the *National Maritime Research Institute*, Japan constructed a new deep-water tank. It is a circular dock (diameter 16 m and depth 5 m) with a deep pit (diameter 6 m and depth 30 m) with a towing carriage, wave maker and current generator.

The developments in both EFD and CFD have increased the demand for instantaneous full-field three-dimensional flow measurements resolved in space and time. In recent years substantial developments have been made in measurement techniques for measuring velocity and vorticity, flow visualisation, pressure, skin friction, free-surface waves, bubbly and cavitating flow, body motion, forces and moments. There is a steady trend from point measurements to distribution measurements such as particle image velocimetry (PIV) and pressure sensitive paint (PSP). New developments continue in optical techniques such as PIV, LDV, and non-contact motion measurements. Techniques like particle tracking velocimetry (PTV) and laser speckle velocimetry (LSV) are now emerging permitting a relatively comprehensive and quantitative measurement of a velocity field in a plane in just one or a few runs; a big step forward compared to previous Pitot or LDV techniques. New manufacturing techniques such as microelectromechanical systems (MEMS) are used to develop new measurement devices such as skin friction sensors.

Experimental approach is undergoing change from routine tests for global variables to detailed measurements for local variables for model development and CFD validation, as design methodology changes from model testing and theory to simulation-based design. Such detailed testing requires that towing tanks utilise advanced modern instrumentation with complete documentation of test conditions, procedures, and uncertainty assessment.

3 Computational Fluid Dynamics approach

In marine CFD the main concern are the problems in hydrodynamics. In the majority of problems being solved, the endeavour is to calculate global pressures and fluid velocity components in a three dimensional space surrounding the submerged portion of a marine vehicle or other structure of interest. In this way, it is possible to further calculate the forces and moments acting on the marine vehicle or offshore structure, whether steady or unsteady. The majority of commercial CFD software tools has been written to solve the more general cases of compressible, viscous, turbulent flows with heat transfer, but may be applied to problems in marine hydrodynamics, so long as the correct choices are made regarding equations of state, fluid properties, and boundary conditions. The presence of the free surface in marine applications provides a major departure from conventional CFD applications and the need to represent this fluid interface accurately presents a considerable challenge.

The application of computational methods in hydrodynamics within the marine industry covers both inviscid and viscous flow calculations. The range of CFD tools available for these classes of problem is broad and varied and their development has followed different paths, with both specialised maritime CFD packages and more general engineering CFD tools being applied to these problems. The need to understand the physics of the problem in hand, the limitations of the equations being used, the basis of the numerical methods employed and the means to get the most accurate and consistent results for the available computing resource, are some of the common challenges faced by the CFD user in maritime sector with the counterparts in other fields of engineering. In marine hydrodynamics in general the methods are used to examine flows which are incompressible, steady and unsteady, laminar and turbulent with or without free surfaces.

For the most fundamental part of marine hydrodynamics, the ship resistance and propulsion, CFD has become increasingly important and is now an indispensable part of the design process. For hull lines design, in practice the CFD applications are mainly concerned with steadily advancing ships on calm water. This corresponds to a numerical simulation of the resistance and propulsion model test. Typically inviscid free-surface methods based on linearised or nonlinear steady free surface boundary conditions have confirmed the usefulness and today are extensively and routinely used as a practical design tool at institutes and still are the preferred solvers at shipyards. Although a model of the final ship design is still tested in a towing tank, the testing sequence and content have changed significantly over the past years. Traditionally, unless the new ship design was close to an experimental series or a known parent ship, the design process incorporated many model tests. This is no longer feasible due to time-to-market requirements from ship-owners and no longer necessary thanks to CFD developments. Combining computer-aided design (CAD) to generate new hull shapes in concert with CFD to analyse these hull shapes allows for rapid design explorations without model testing. CFD allows the preselection of the most promising design and then often only one or two models are actually tested to validate the intended performance features in the design and to get a power prediction accepted in practice as highly accurate. As a consequence of this practice, model tests for shipyard customers have declined considerably since the 1980s [1].

The most important methods in practice are Laplace solvers for potential flows and Navier-Stokes solvers for viscous flows. The Laplace equation is the fundamental equation for potential flow and typically free-surface codes are based on boundary element methods (BEM), also called panel methods. These codes are used to analyse the ship forebody; especially the interaction of bulbous bow and forward shoulder because disregarding viscosity introduces considerable errors in the afterbody. The Navier-Stokes equations fully describe the flow about a ship. However, they cannot be solved analytically or numerically for real ship geometries. The large Reynolds number makes the flow turbulent along the hull and eddies of widely different scales have to be resolved. For a ship advancing at 10 m/s, the smallest eddies are of the order of 1 μm with a period of 10^{-5} s. The computational domain is of the order of 10^3 m^3, and the required time to integrate over the largest eddies is of the order of 10 s and the space-time discretisation of this problem calls for an extremely large number of computational cells [2]. Therefore Reynolds-averaged Navier-Stokes equations (RANSE) are used to solve the problem. These equations relate the turbulent fluctuations with the time-averaged velocity components and the relationship can only be supplied by semi-empirical theories in a turbulence model. The modelling of turbulence is possibly the most actively researched subject in CFD.

BEM cannot be used to solve RANSE since the whole fluid domain should be discretised. Therefore the field methods are required and both finite difference methods (FDM) and finite volume methods (FVM) are used. While FDMs lose popularity and FVMs gain popularity, FDMs give in many cases results of comparable quality. Most commercial RANSE solvers today are based on FVMs and the fundamentals can be found in [3]. Finite element methods (FEM) dominate structural analysis and for marine hydrodynamics they play only a minor role.

The BEMs are expected to remain the workhorse of the industry until at least the 2020s [1]. This global decomposition changes as RANSE computation drifts more into commercial applications and for the future it is foreseen the massive use of RANSE codes. The trend in using RANSE solvers for real ship computation at model scale has become stronger and better CFD codes are ready to be used for design purposes. But the majority of RANSE solutions are still for model scale and the number of codes that predict full-scale viscous flows still seems to be limited. So far, most RANSE computations for practical ship design were just double-body calculations, in which the effect of the wavy surface on the viscous flow and of the viscous flow on the wave-making were disregarded, while the current status of RANSE codes incorporate this interaction and free-surface boundary conditions are solved. The prediction of the total resistance is still a difficult task and effort should be performed in reducing the numerical uncertainty. Results of a RANSE computation with a good visualisation provide much more detail and more possibilities for inspection than a model tests and in less time, and are found very helpful for design improvements as well as for the orientation of appendages. An overview of commercial CFD software tools for these applications can be found in [4]. The use of integrated CAD systems and CFD solvers in the design of ships is one of the key issues in the application of CFD for practical design of ships. There is considerable interest in the use of optimisation techniques coupled with CFD in order to link directly the process of hydrodynamic analysis and design. The techniques have been developed primarily for implementation with respect to the wave resistance problem, with some simple empirical calculation of viscous resistance. It appears that tools for design optimisation are maturing, and that it is feasible to use some of these techniques in practice, although the computational effort required puts much of this work beyond the capability of a typical design office.

The subject of unsteady flows covers two areas of interest to a hydrodynamicist. The first involves the response of the vessel in waves, and includes both its rigid body response and structural behaviour. The second concerns ship manoeuvring and the ability to simulate or predict the flow around and fluid forces and moments acting on the ship or its control surfaces as the result of course changes.

The application of CFD to predict ship motions and hydrodynamic loads imposed on the hull structure is relatively new, and it remains the case that full unsteady RANSE solutions of ships in waves are rare and unvalidated. Traditional approaches to calculating ship motions and loads, developed in the late 1950s and early 1960s were based upon the concepts of classical hydrodynamics. These methods led to so-called “strip theory” computer programs, working primarily in the frequency domain, to give solutions of the linearised two degrees of freedom equations of motion (coupled pitch and heave). Later, the development of three-dimensional approaches for wave diffraction problem in the frequency domain provided further generalisation, such that all six degrees of freedom of ship motions could be addressed. These techniques produce results, which are efficient and sufficiently accurate for engineering purposes, over a wide range of vessel types, speeds and environmental conditions. The 1990s saw the advent of Rankine panel methods for seakeeping. In the frequency domain, quasi-steady BEMs compute the forces and motions of a ship in regular waves. However, time-domain methods were more versatile and were turned first into commercial flow codes, although development on time-domain codes started several years later. The approaches are similar to those used for the steady wave-resistance problem, but far less mature. The probabilistic approaches to load prediction in spectral seas, aimed at calculating long term loading statistics, are a key requirement in naval architecture. The CFD simulations are not well suited to such applications and therefore there has not been a significant driver behind the development of RANSE based CFD for basic seakeeping calculations. There are, however, two general areas where CFD based simulations methods have received attention in seakeeping, and these are large vessel motions and extreme loads in steep waves and slamming, for which the linear theories break down, and roll motions, damping and the actions of active fin stabilizers and bilge keels.

For the first of these, the CFD has been applied to both quantifying local loads and flow phenomena, and complete non-linear ship motions. The former particularly includes slamming, water impact, and the shipping of green water, to which unsteady RANSE methods have been applied. The prediction of large-scale ship motions is essential to the calculation of extreme loads and appears to have been approached mainly through the use of non-linear inviscid and potential flow methods. The other area where CFD is applied to seakeeping and loads prediction is that associated with the simulation of large motions and steep waves. These methods seek to simulate the behaviour of the vessel by calculating the complete fluid loading and its distribution over the instantaneous wetted surface at each instant. Even with current computer technology, this represents a considerable challenge. The current state of the art therefore is to carry out such calculations using inviscid flow models, and RANSE calculations of the complete ship seakeeping problem remain a considerable challenge. Numerous two-dimensional simulations of the roll motions of ship sections have been carried out with the aim of predicting the viscous contribution to roll damping. Potential flow BEM approaches for the non-linear simulation of motions and loads are in active use for conventional hull forms, and commercial tools have become available. The presence of forward speed modifies the associated wave field in a very fundamental way, which consequently introduces the huge difficulties in the evaluation of the corresponding potentials which are used in the BEM solvers. Linear diffraction methods and strip theory are still in extensive use however in industry. RANSE methods remain for research only. High-speed craft or other novel hull forms appear not to have been addressed directly. Both BEM potential flow and RANSE calculation methods have been used for studies of detailed elements, such as slam loading or roll damping in 2D. For the future, it would appear unlikely that unsteady RANSE methodologies will be deemed suitable for general use in design and analysis.

Aspects of manoeuvring properties of ships gain in importance, as public opinion and legislation are more sensitive concerning safety issues after spectacular accidents of tankers and ferries. IMO regulations concerning the (documented) manoeuvrability of ships increased the demand for CFD methods in this field. Model tests as an alternative method are expensive and time consuming. The calculation of manoeuvring forces and moments using CFD techniques has made considerable advances in recent years with the ever-increasing computing power available making unsteady flow simulations more practical. Both potential flow solvers and RANSE techniques have been developed. In addition, researchers have chosen to study both complete ship hull flows and model the behaviour of individual appendages. The important forces and moments in manoeuvring arise from inertial, viscous and lifting effects. The manoeuvring derivatives are even now mostly derived by carrying out specific forms of experiment in which a model is made to carry specific motions in a towing tank, and the forces and moments acting on it are measured. Such experiments are quite involved however, since there are a large number of manoeuvring derivatives required for the equations of motion, and frequently recourse is made to empirical formula and past databases for certain of the terms. Both BEM and field methods have been employed for selected manoeuvring problems. Potential flow methods are only appropriate for the modelling of inertial and lifting effects in manoeuvring. The former is straightforward, since classical hydrodynamics deals easily with the computation of added mass or rotational inertia effects. The prediction of lift presents more of a problem. The potential flow methods can be more easily verified for slender or classical lifting geometries, where the results are less sensitive to the absolute accuracy of the resulting vortex field. This is not the case for full hull forms however, where significant amounts of tuning are required in order to achieve reasonable agreement with measured manoeuvring forces and moments. One way to remove these difficulties in the prediction of manoeuvring forces is through the application of RANSE methods, which in principle should capture the complete details of the flow field and vortex structures, and their influence on the forces and moments on the hull. The main difficulties in achieving such improvements lie with the ability of the RANSE techniques to predict flow separation. This would therefore appear to be a promising area for the development of RANSE solvers in marine hydrodynamics, although as ever, there is some need for further validation, mesh and turbulence modelling sensitivity studies, before confidence and trust can be established. Both RANSE and potential flow based methods have been used in research and practical design of rudders, thrusters and podded propulsion systems. The use of unsteady RANSE methods for hull/propeller/rudder interaction is well established, but requires considerable care by users. The computational methods and best practice (in the sense of grid resolution, choice of turbulence models etc.) are similar to those developed for aerodynamic applications and similar rotating flows, and so there is considerable opportunity for validation. Therefore the technology is in a position where it is ready to be exploited in the design and optimisation process.

The main emphasis in offshore floating systems is the calculation of wave loads on stationary or very slow moving offshore floating production systems. Loads on these structures are affected by wave diffraction, and hence three-dimensional panel methods were developed to calculate these loads. Viscous effects have a role also, to varying degrees depending upon the type of systems and its location. In potential flow diffraction modelling the linear, frequency domain diffraction analysis is regarded as a mature and well exploited computational tool. Best practice is well understood and computer programs well validated. The next level up is second order diffraction analysis. Additional potentials due to sum and difference frequency components are evaluated using extensions of first order techniques. Here too, the theory and modelling techniques are sufficiently mature for the improved calculation of low frequency motions (drift) and high frequency effects such as tether springing. Improved estimates of air-gap as the result of closer modelling of near-field diffraction and interference effects are also possible. Fully non-linear time domain diffraction analysis is also possible, and some models exist primarily for research purposes.

Wave run-up and green water loading calculations provide the ultimate challenge in free surface dynamics, not least as the result of the severe wave breaking and spray forming effects that occur. Potential flow models are unsuitable in such cases and, though less well mathematically formulated, volume of fluid (VOF) simulation technique appears to offer the only feasible approach at the present time. Calculations of vortex shedding on circular cylinders in steady flows is regarded as a standard "benchmark" in CFD, with the objective of reproducing the Karman vortex street observed in experiments. It is remarkable nevertheless that many commercial CFD codes are unable to adequately model this basic phenomenon unless very great care is taken with the numerical discretisation scheme and near wall and turbulence modelling.

In the domain of ship propulsors, inviscid flow methods have long been used in propeller design as a standard tool yielding information comparable to experiments. Lifting-surface methods and BEM are equally popular. Lifting-surface methods (quasi-continuous method, vortex-lattice method) allow the three-dimensional modelling of the propeller. They discretise the mean camber surface of the propeller blade by individual vortex panels. In addition, the free vortices are modelled by elements of given strength. Lifting surface methods are well established for the evaluation of forces and moments of marine propulsors. Efficiency of lifting surface methods having short computing times makes the methods popular for iterative evaluation of propeller performance, such as optimisation of propeller geometry. BEMs represent an improvement concerning the treatment and modelling of the geometry and are most widely used type of panel code for conventional propellers. Panel methods are widely used for the performance prediction with an increasing level of confidence, while requiring reasonable computing time. Panel methods have been extended to problems of the unsteady performance and cavitation prediction of propellers. The recent improvements in panel method codes have resulted in good agreement between the measured propeller and predicted performance and pressure distributions on the blade for most types of propellers. For unconventional propellers, both steady and unsteady vortex lattice methods are capable of predicting the overall performance of a contra-rotating propeller set at the design point with at least satisfactory, and in some cases, with good accuracy. All published end plate propeller design procedures need empirical corrections in order to obtain the specified design performance. Panel methods are helpful in the design of propeller boss cap fins, but still require further developments to obtain accurate performance predictions. The existing design methods provide practical design methods for propellers with pre-swirl vanes. In the last decade the panel method codes have been extended to treat quite complicated interaction flows such as podded propulsor and cavitating propeller-rudder interaction flow. In the last few years substantial advances have been made in the application of RANSE codes to ship propulsor flows. Today the number of cells required for such calculations limits more widespread use of RANSE to interaction studies in shipyard design work. In recent years two-phase and cavity models have been added to the leading commercial RANSE codes. RANSE codes are capable of predicting accurately the propeller open water performance and pressure distributions on the blade at the design advance number. The accuracy of performance prediction for smaller off-design advance numbers is good or acceptable. The main features and the main variables of the flow field, streamlines and boundary layer are generally well predicted. Unsteady vortex lattice and panel methods are the main CFD tools used for predicting unsteady cavitation on propulsors and propulsor induced hull excitation forces. The recent improvements in panel method codes have resulted in good agreement between the measured propeller and predicted performance and pressure distributions on the blade for most types of propellers.

Less frequently found applications of CFD in marine hydrodynamics include computation of the airflow around the upper hull and superstructure of ships and offshore platforms, wind loads for ships with large superstructures, tracing of funnel smoke for passenger vessels etc. Interior flow problems like flows in roll damping tanks, sloshing are also treated in marine hydrodynamics. Table 1 summarises an assessment of the maturity of various CFD applications in marine hydrodynamics [1].

Table 1 **Maturity of CFD applications**
Tablica 1 **Zrelost za primjenu modela RDF**

	Inviscid	Viscous
Resistance test	• • •	••
Propulsion test	-	••
Ship seakeeping	••	•
Manoeuvring	•	•
Offshore	•••	-
Propeller	••••	•
Others	-	•
Scale: “-“ not applicable, no applications known; “••••” very mature;		

The marine and offshore industry shows the wide-ranging interests in the use of marine CFD and recognises the versatility of CFD in solving practical engineering problems. The industry suggests the following general topics that should be considered for future research in CFD [5]:

- The integration of tools and techniques across all aspects of the analysis process (potential flow and RANSE, rigid body and structural dynamics);
- The improvement of solver speed and accuracy;
- Full scale validation of CFD through improved measurement techniques and equipment;
- The introduction of tools for optimisation of form and performance.

4 CFD challenges in marine applications

The growth in the use of CFD codes and the trend for them to become rich packages with lots of alternative modelling options steadily increases the risk of user errors. This trend is reinforced by the ease of use of modern computer codes with simple graphical user interfaces making them available for inexperienced users. Although efforts are taken to simplify the usage of CFD codes, careful training with realistic exercises should still be considered as the starting point of any CFD user. The theoretical part of the training should focus on fundamental modelling features, their underlying assumptions and their limitations. Unlike linear finite element stress analysis, CFD still requires expertly trained users for good results. In situations where non-experienced users have to be used, some restriction on their freedom to adjust critical parameters might be advisable, and they should be limited to simulations of routine types. Depending on the CFD software, additional training on grid generation is advisable. A CFD user for non-routine applications should have good training and knowledge in classical fluid mechanics, broad understanding of numerical methods, and detailed knowledge of the application being examined. This means that they will be able to understand the limitations of the models used (e.g. turbulence, radiation, etc.). The training and education requirement for more routine applications can be less stringent, provided that clear guidelines or procedures have been established for the use of the code being used. An example of a routine application would be the simulation of a standard component in a design environment where many previous designs have been calculated and only relatively small changes in geometries and boundary conditions occur. In both routine and non-routine applications, training on the use of the specific CFD code with the solution of realistic exercises is needed.

The deficiencies or inaccuracies of CFD simulations can be related to a wide variety of errors and uncertainties. The process of performing a CFD calculation is itself complex and requires the competence in different activities. These typically include: definition of the problem; selection of the solution strategy; development of the computational model; analysis and interpretation of the results. All of these steps are potentially error prone or subject to some degree of uncertainty. An error can be defined as a recognisable deficiency that is not due to the lack of knowledge, and uncertainty as a potential deficiency that is due to the lack of knowledge. To be clearer, an error is something that can be removed with appropriate care, effort and resources; whereas an uncertainty cannot be removed as it is rooted in the lack of knowledge.

There is no universally excepted means of identifying or classifying errors, which can range from human or user errors to inadequacies in the modelling strategy and model equations. However, the following classification based on several different sources of error and uncertainty can be accepted [6]:

- Model error and uncertainties (errors due to the difference between the reality and the exact solution of the proposed model equations);
- Discretisation or numerical error (errors that arise due to the difference between the exact solution of the modelled equations and a numerical solution on a grid with a finite number of grid points);
- Iteration or convergence error (errors that arise due to the difference between a fully converged solution on a finite number of grid points and a solution that is not fully converged);
- Round-off error (errors that arise due to the fact that the difference between two values of a parameter during some iterative scheme is below the machine accuracy of the computer);
- Application uncertainties (defined as inaccuracy that arises because the application is complex and precise data needed for the simulation are not available);
- User errors (errors that arise due to mistakes and carelessness of the user);
- Code errors (errors due to bugs in the software, unintended programming errors in the implementation of models or compiler errors on the computer hardware being used).

The user remains the main area of uncertainty in application of marine CFD, and inconsistent practices the main source of error and inconsistency. It is likely therefore that CFD will remain a specialist discipline in design analysis rather than become a mainstream process for shipbuilders, naval architects and offshore engineers.

In discussions of CFD errors and uncertainties it is useful to make some clear distinctions between the meaning of the terms validation, verification and calibration. Verification of a numerical model means checking that the computer program correctly represents the continuous mathematical model on which it is based. On the other hand validation is the demonstration that the verified computer program applied to such models is an adequate representation of the physical reality. Validation is a broader activity, which includes verification and comparison with other results (analytical, experimental or numerical), Figure 3 [7].

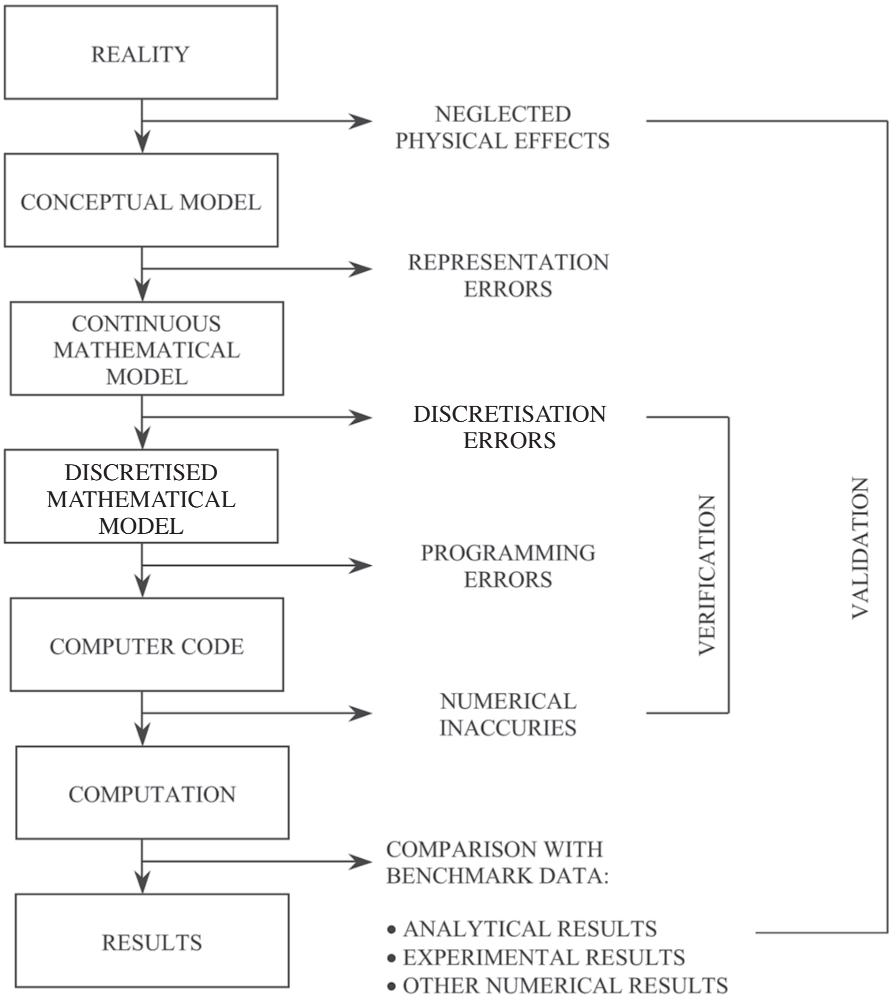
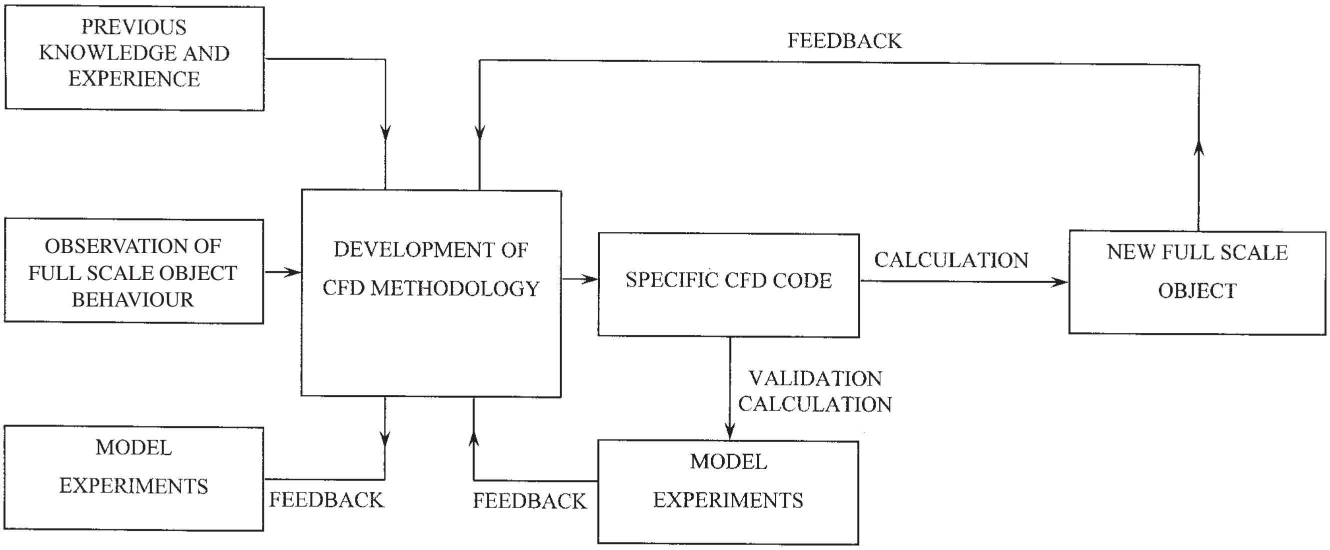


Figure 3 **The verification and validation process in CFD**
Slika 3 **Postupak provjere i ocjene valjanosti modela u RDF**

Figure 4 **The place of physical experiments in the CFD development process**
Slika 4 **Mjesto fizičkog pokusa u procesu razvoja modela RDF**



Calibration is a procedure to assess the ability of a CFD code to predict global quantities of interest for specific geometries of engineering design interest. The ITTC has seen the uncertainty analyses for CFD as a natural extension of its earlier activities and much work has been undertaken in that field [8, 9]. The place of physical experiments in the CFD development process is presented in Figure 4 [10].

According to [9], the types of model tests carried out in experimental facilities have undergone a considerable expansion, and today can be classified as follows:

- Fundamental tests, aimed at giving insight into basic phenomena;
- Correlation tests, meant to validate a computational method;
- Systematic tests, to collect hydrodynamic data for a systematic parameter variation, e.g., to be incorporated in prediction software in the form of empirical coefficients;
- Feasibility tests, to confirm estimates of the behaviour of new ship concepts before the concept is further developed;
- Design tests, aimed at producing data specifically relevant to a particular design or operation.

In the past, the emphasis for most towing tanks was on design testing. The growing importance of computational methods requires an increasing amount of validation testing. So far, experimental data used for validation in many cases concern classical cases and there is a clear need for more validation cases, more relevant to modern marine vehicles and other marine structure types. Fine agreement in predicted global data does not mean that the theoretical approach is correct; bad agreement does not tell us where the theory is wrong. Hydrodynamic experiments, such as measurement of fluid velocity and pressure distribution and flow visualisation will be necessary to prove the validity and the range of applicability of the numerical model used or to pinpoint inaccuracies in the modelling. This means that the increasing emphasis on validation testing should ideally involve a shift in the type of experiments requiring special instrumentation. However, also for design testing and hull form optimisation, experimental determination or observation of detailed flow features may create new possibilities. One of the fundamental drawbacks of conventional model tests is the fact that the flow character is hard to observe. Tuft tests, paint tests, and observation of the wave pattern produce an incomplete picture that could only be complemented by time consuming velocity and pressure measurements using Pitot tubes or LDV. It seems, however, quite possible that modern experimental techniques will allow a much more efficient experimental determination of the flow field than has ever been possible before.

Marine hydrodynamics is very much in motion with new types of marine vehicle and offshore structure forms, new design tools, and new techniques for experiments and computations emerging. Higher demands on marine problems, a shift towards more complicated model tests, more sophisticated experimental techniques, and complicated computational problems pose significant challenges for marine hydrodynamics institutes. The increasing role of computational methods in marine problems and optimisation imply that availability of a towing tank could become a less decisive advantage and marine vehicle design and evaluation could move away from classical institutes. On the other hand, the combined and integrated use of numerical and experimental marine hydrodynamics together with a large amount of know-how collected continues to be a major asset. However, an active position is desired to continuously enrich this with the new techniques becoming available. It seems reasonable that the activities of towing tanks change from routine tests to more sophisticated tests oriented to local measurements instead of global measurements. The development of new experimental techniques tends to strengthen again the role of model testing. It could well be useful for towing tanks to have an active role in developing and applying these tools. It is important that the personnel are knowledgeable not only about the operation of the facility or code, but also with respect to fundamental assumptions and area of application and limitations. With respect to experimental facilities, this involves not only the knowledge and experience of the basin technicians, but also of the complete chain of personnel. With respect to the increasing application of more sophisticated numerical methods, management would have to take steps to ensure that staff has sufficient expertise, not necessarily regarding software, but at least to be able to meaningfully communicate with those who do so on their behalf. If the management of towing tanks makes a conscious effort to come to decisions to keep improving the model testing and scientific staff as well as the whole chain of personnel, it will be possible that the activities of towing tanks will increase in the future to answer all the new needs of marine construction [8].

5 Conclusion

Marine hydrodynamics is undergoing rapid advancement due to new developments in marine vehicle and offshore structure concepts and design tools, in CFD methods and their use for design and in sophisticated EFD measurement systems and their use in more complicated local-flow model tests. The marine hydrodynamics institutes significantly expand their activities beyond routine testing and use the accumulated knowledge to include expertise in complementary CFD and EFD. After a hundred years of reliance on experiments for the detailed design of marine vehicles and other marine structures, the marine industry enters a period when numerical models will be able to provide a credible and more effective alternative for a wide range of hydrodynamic design problems. This does not, however, remove the need for model testing and collection of data at full scale. Only through an integrated activity of numerical testing and physical experiments is it possible to validate the computational models. To achieve all this effectively, there must be a synergy between the experimentalist and the numerical analyst.

Prof. W.R. Eatock Taylor said [10]: “We have progressed far since Froude applied the scientific method to playing with boats. We must continue to be active players, now using both real and virtual models to improve our understanding and thereby to optimise our designs”.

References

[1] BERTRAM, V.: “Practical Ship Hydrodynamics”, Butterworth-Heinemann, Oxford, 2000.

[2] LARSSON, L, BROBERG, L., KIM, K.J., ZHANG, D.H.: “A Method for Resistance and Flow Prediction in Ship Design”, SNAME Transactions, Vol. 98, 1990, p. 495-535.

[3] FERZIGER, J.H., PERIC, M.: “Computational Methods for Fluid Dynamics”, Springer Verlag, 2001.

[4] …: http://www.bodrum-bodrum.com/vorteks/arsenal/cfdcodes.htm, October 2005.

[5] …: “MARNET CFD Final Report and Review of the State-of-the-Art in the Application of CFD in the Maritime and Offshore Industries”, WS Atkins Consultants LTD & Members of the Network Steering Committee, 2003.

[6] …: “Best Practice Guidelines for Marine Applications of Computational Fluid Dynamics”, WS Atkins Consultants LTD & Members of the Network Steering Committee, 2002.

[7] …: “Proceedings of the 19th ITTC Conference”, Madrid, 1990.

[8] …: “Proceedings of the 22nd ITTC Conference”, Seoul and Shanghai, 1999.

[9] …: “Proceedings of the 23rd ITTC Conference”, Venice, 2002.

[10] TAYLOR, E.W.R.: “From Froude to the Supercomputer: Modelling Marine Vehicles in Waves”, The Royal Academy of Engineering, London, 1992.