Implications of CFD applications: towards a new strategy of ship hydrodynamic design

G. Trincas

Department of Naval Architecture, University of Trieste, I-34127 Trieste, Italy

Abstract

The increasing efficiency and reliability associated with application of CFD codes in different ship hydrodynamic domains is discussed. Uncertainty analysis as the standard technique for experimental accuracy assessment and CFD validation is advocated. Application of CFD to bring the model-to-ship scaling techniques from the present empirical methods to a more theoretical basis is illustrated. Integration of CFD codes into the hydrodynamic design leading to a complete revolution in the ship design strategy by introduction of a multicriterial decision-making process, is summarized.

Introduction

During the last century, entering into operation of new ship concepts put a growing pressure in the introduction of more accurate prediction methods that produced designs of higher hydrodynamic performance. For many decades, at early design stages rules of thumb and empirical methods based on statistics of previous performance predictions were used, which were then coupled with model experiments on the final design only.

The development of powerful computers and advanced numerical codes has made it possible to describe some complex flow phenomena by CFD methods, which have begun to play a major role for applications in ship hydrodynamic analysis and design, progressing from academic research codes to practical design tools. Although the numerical methods used in this context are peculiar to equations ranging from the Euler equations for inviscid flows to the complete Reynolds-Averaged Navier Stokes (RANS) equations for turbulent incompressible flows, CFD applications in the shipbuilding industry are still less developed than in other industrial fields. The related difficulties are partly due to partial inadequacy to handle such complex phenomena like viscous drag and free-surface effects, but mostly because of conservative naval architects' refusing to introduce new design approaches which could incorporate at best CFD advanced tools.

A clear understanding of which are the actual and future CFD capabilities in the
sense of reliable applications in ship design, requires a new strategy of the basic
design process moving to the introduction of what may be named Hydrodynamic
Aided Ship Design (HASD). Apart from intrinsic difficulties both theoretical and
numerical relevant to some hydrodynamic domain, a real revolution is going to
take place in ship hydrodynamic design leading to more efficient ship, harvesting
the fruits of more rational design procedures.

Numerical ship hydrodynamics

Programs based on simplified Navier-Stokes equations are widely used in many
important areas of ship design such as wave resistance, propeller performance,
and seakeeping. Many examples of their successful application proved the real
usefulness in making the design procedure more rational and in confining the
model tests to those which are strictly necessary to perform final improvements.
The potential flow computer codes have progressed as far as possible also because
the free surface problem and the consequent theoretical modelling of the outer
inviscid flow is unique to hydrodynamics. Nevertheless, application of CFD to
the ship resistance problem is a different issue, mainly as regards viscous flow. In
this respect two main problems are encountered which are not so relevant or do
not appear in other engineering fields, i.e., high Reynolds numbers and free-water
surface effects. The full Navier-Stokes equations may be solved only for special
cases.

Resistance

Current practice makes Froude's method still prevailing over numerical ship re-
sistance prediction methodology. Resistance predictions without model tests are
adequate only for hull forms well represented by existing model series, while
selection of correlation allowance is one of experience. However, two different
approaches have been devised to predict the flow and resistance components of
ships and marine vehicles by CFD.
The first approach, the so-called multi-domain zonal method, divides the flow do-
main into three flow zones of different nature, namely, the free-surface potential
flow for wave-making resistance evaluation [1], the thin boundary layer [2] and
the turbulent boundary layer around the ship stern treated by parabolic equa-
tions for the viscous flow and wake modelling [3]. The aforesaid methods are
then integrated, though there is great demand for more fundamental studies in
the interaction of these flow zones.
The second approach considers the whole domain simultaneously by modelling
the flow through fully elliptic RANS equations written in non-orthogonal curvi-
linear body fitted coordinate system [4]. The generality of the method is one
of the important trends of fluid dynamics research, thus allowing the scientific
community of naval architects to interact with different fields of hydrodynamic
research. The simulation of fluid flow phenomena demands numerical solution
of the Navier-Stokes equations involving terms which describe the convection,
Computational Methods and Experimental Measurements 405
diffusion and any sources present. The numerical solutions require interpolation assumptions for the variation of the fluid properties and their gradients between discrete points on a computational grid that covers the fluid domain.

In the area of viscous resistance CFD is becoming increasingly important, although there is still a heavy reliance on experimental results because viscous flow cannot yet be adequately modelled as confirmed by a recent experimental work too [5]. The problems of the so-called false diffusion and turbulence structure modelling around the stern require further research. Defects in the turbulence model have main responsibility for the larger part of discrepancies observed in the wake predictions of ship-like bodies and for overpredicting the total viscous resistance coefficient of submerged bodies of revolution too [6]. As regards the approximation of the convection terms in the Navier-Stokes equations, important improvements of solvers for a system of elliptic differential equations based on finite-difference algorithms, have been carried out in recent years [7, 8]. The physical criteria of reciprocity of fluxes at cell faces, conservation, and consideration of the local direction of flow are to be satisfied together in order to converge to physically realistic solution. Although the developed methods provide the account of main features of stern flow formation, most of the existing schemes fail to satisfy all the above criteria.

As thick boundary layer is characterized by strong viscous-inviscid interaction, methods capable of solving both viscous flows and nonlinear free-surface deformation simultaneously have been developed in recent years, by moving the boundary-fitted grid system according to the wave motion [9].

Propeller analysis methods

The flow around a propeller can be dealt with the potential theory too, except in the boundary layer surrounding the screw profiles and its wake. Three-dimensional lifting surface and surface panel methods have been proved quite powerful in deriving blade surface pressure distributions and analysing propeller performance by modelling the sheet cavitation dynamics [10].

Theory allows to control cavitation with good accuracy in propeller design for most cavitation types. However, cloud cavitation with tip vortex can not be predicted well so that full-scale cavitation prediction still depends on model tests, even if scale effects on vortex cavitation make reliable predictions difficult. No Navier-Stokes application exists for non-uniform wake-adapted propellers.

Seakeeping

On the seakeeping field computational tools have been successfully validated and used since many years and many codes are incorporated into the ship preliminary design process. This development started with the application of the linear two-dimensional strip theory for the prediction of ship motions and structural responses to impact-type wave loads, giving remarkably good results even for low ship length over beam ratios and for high Froude numbers [11]. It continued with the development of three-dimensional boundary integral equation methods for
the prediction of motions and wave loads on large volume offshore structures at zero forward speed [12]. These methods were extended as to cover low frequency second order effects and low forward speed effects.

During recent years, new time-domain nonlinear ship-motion and sea-load codes have been developed to evaluate large amplitude motions and near capsize conditions. These three-dimensional programs satisfy the exact instantaneous boundary condition on the body while the free-surface boundary condition is usually linearized [13]. It is believed that this kind of programs will soon become a reliable design tool. Excluding viscous phenomena, such programs generally combine the results of linear seakeeping programs for diffracted and radiated wave forces with a nonlinear approach to oncoming wave forces. The important viscous flow phenomena related to roll damping as well as to lift and drag contributions to sway force and yaw moment, are included using appropriate semi-empirical expressions.

**Manoeuvrability**

The prospects for CFD applications is still worst in the field of controllability. The best approach now available to study the manoeuvring behaviour of ships utilizes time-domain simulations by means of a mathematical model based on the differential equation system which describes the ship dynamics. Although a few of the linear manoeuvring derivatives, such as added mass and linear damping terms, can be obtained by means of potential theory calculations, the applicability of theoretical flow codes is very limited so that capability of CFD to provide reliable design tools is at present very remote. Viscous phenomena and flow separation play an important role too, aggravated by the occurrence of considerable cross flow on hull with strong vortex shedding. Control performance is still predicted mostly using either model tests or equations of motions with coefficients obtained semi-empirically from the results of model tests on similar hull forms.

**Validation problems**

The advantage of CFD is that it is in general faster and thus cheaper than physical testing, but there are still uncertainties of how reliable and accurate the results are. Although CFD applications in different ship hydrodynamics fields are increasing, much effort is to be done before the 'numerical towing tank' can become reality. Therefore, a combination of numerical and physical testing is still most cost-effective to get sufficiently accurate results also for prediction of full-scale behaviour.

The validation of a CFD prediction can be done after a process of code verification as well as numerical and grid sensitivity analyses have been completed, although the comparison of computed and experimental results is often a difficult task, since experimental data are subject to errors and inaccuracies too.
Synergy between CFD and experiments

The towing tank has been the analogic tool utilized by naval architects for many years to optimize hull forms. Notwithstanding a number of drawbacks (high costs, evaluation of scale effects, long lead times), model testing is still an essential element in the hydrodynamic design process because not all flow conditions can be simulated accurately by CFD. There is, however, a shift of effort and costs from the routine design testing towards research and development. On the other hand, the numerical approach can assist in understanding flow phenomena and physics which cannot be recognized directly from model experiments.

Ship hydrodynamics calculations and experiments have progressed in separate directions. But application of computational methods is significantly affecting the role of model experiments to serve validation purposes, creating a need for new types of more sophisticated tests and special experimental techniques. Among the others, there are still some main features to consider before CFD tools may be introduced into the ship hydrodynamic design extensively. They are verification of the computer programs and validation of the predictions. Verification refers to numerical accuracy by checking whether there are no errors in the discretized model and whether the coding is free of bugs, involving comparison of numerical results with exact analytical solutions. Validation refers to how adequately the basic equations represent the physical reality, implying comparison between computations and full-scale results too. Also the accuracy of experimental results has to be addressed rigorously by a comprehensive uncertainty analysis through statistical techniques. This accuracy is partly determined by the measurement techniques and by the reproducibility of the experiments. The problem remains of how an uncertainty analysis might be identified and applied to numerical predictions.

Validation from full scale

The impact of the coupling of CFD and experiments can be most beneficial to correlate to full-scale with some confidence, provided the flow regime does not change with respect to model scale. The most complicated and controverted element in the prediction of propulsive properties of a ship on the basis of model tests is the scaling of the local characteristics of the flow as well as of the wake and thrust-deduction fractions. There is a great number of works devoted to the problem, but the semi-empirical methods used for Reynolds number scaling of ship wakes can not adequately take into account all the complexities of ship stern flows. Hence, the weakness of the current extrapolation method must be emphasized. The main difficulty in applying a numerical method for the solution of the RANS equations at Reynolds numbers of the order of a billion, is to capture the dynamics of a viscous and turbulent flow, to maintain accuracy in the resolution of the increasing gradients of velocity and turbulence parameters, unless the vorticity that is generated within the wall layers is accurately calculated. Practically, this requires finer and finer grids in regions close to walls, and that translates into many grid points, large computers, and increasing computing
times. For the time being, the incremental velocity method utilized to determine the full-scale velocity profiles of a body of revolution, can be applied for extrapolation from lower Reynolds number nominal wakes to full-scale propulsion design [14]. In any case, the wake scaling problem is still unsolved, requiring further correlation work.

Feedback from full-scale should provide the ultimate validation of both computational and physical models. Accurate and reliable quantitative information at ship scale are scarce because measurements on real ships are expensive, very difficult to perform and not too popular with ship operators. With modern hardware, it is becoming feasible to put equipment on board to measure full-scale data efficiently, thus yielding a great impact on the validation of CFD design tools.

**Impact of CFD on ship hydrodynamic design**

Ship hydrodynamic design is aimed at fulfilling a number of requirements to be met by the ship in compliance with particular criteria of performance effectiveness. CFD is altering tremendously the manner in which ship design is performed. For the time being, introduction of CFD has increased the speed with which the design iterations are accomplished and is reducing the number of these iterations. Introduction of new numerical codes into the design offices requires improved computer-aided systems. The effect of hull changes as a consequence of correctly predicted flows (inverse problem) impacts other areas such as structures, vibrations, powering, construction, economics, which have to be considered simultaneously through multiattribute decision-making (MADM) approaches for a multi-disciplinary designs [15].

To facilitate application of CFD codes, pre-processing software has to be developed to automatize most of the sometimes laborious input preparation by graphic-interactive tools. CFD codes require geometric input in the form of specially arranged sets of points, by generating automatically a complete, accurate, and unique three-dimensional model to all analysis methods. Present-day results confirm that if ship geometry descriptors are coupled to fast panelization, gridding and flow analysis codes, the resulting HASD system provides a capability for realistic flow calculations within real time relevant to design at an early stage. Nowadays, use of CFD tools is often hindered by a lack of good integration between the above mentioned codes. The transfer of data from one program to another is often tedious and source of errors. The capability to pass hull form data between surface modelling and CFD programs, if coupled with MADM procedures, should allow rapid and efficient comparisons among alternative hulls. Finally, stored computational models may form the basis for input to the flow codes, helping the designer to build the necessary input in a proper way starting from a ship reasonably similar to his own project.

The shell of the HASD system is shown in Figure 1. This structure implies storage of the statistical material accumulated concerning the hull forms and tests of improved ship lines, also for validation procedures of numerical estimations.
Within the framework of this structure information necessary for taking decisions based on the results of computations (synthesis process) form the "attribute data base", whereas data to be transferred to other functional tasks of the design process form the "target data base" (analysis process). When solving the problem of hull form multiattribute optimization under a set of criteria and constraints, use is made of the method of inverse mapping from target space to design space, i.e. calculation of the most appropriate values of design variables for given aspiration levels of targets [15]. The generation of the so-called Pareto sets, i.e. hull form designs that can not be simultaneously improved to satisfy all the criteria, allows to obtain as optimum the trade-off results through the synthesis process which calculates attribute function values for given values of design variables.

All things considered, the strategic goal remains use of CFD tools in the crucial phase of ship's hull form definition, if CFD is wanted to become a key support to ship design. As all technical solutions in the initial ship hydrodynamic design stages are to a greater extent decisive for the economic efficiency of the ship, it is necessary to develop a design strategy making easier to the utmost degree the best technical decisions. The proposed multicriterial decision-making support system (MCDSS) conceived to this purpose, is illustrated in the form of a flow-diagram (Fig. 2). To dominate the design problem definition, the interactive structure of MCDSS allows description of multicriterial models on the basis of heuristic decisor preferences and strategy as well as selection of goals, criteria and attributes with corresponding weights. This structure is suitable both to concept and preliminary ship hydrodynamic design. Full description of MCDSS components is reported in Ref. [16], where a relational model is advocated for the data base. The relational data base is a set of various elements and relations which are expressed vaguely. The vague elements can be modelled by fuzzy sets and the relations by fuzzy relations introducing into the data base the membership grades interactively. Among the others, application of MCDSS will allow very fast evaluation of different design criteria on performance of feasible competitive designs up to activity for tendering documentation. Hence, ship design will be no more a sequence of iterative procedures but a real parallel process devoted to designer creativity for innovative ship concepts in combination with high efficiency products.

Conclusions

The chances of numerically computing the hydrodynamic characteristics of a ship have increased in the last decade, providing the maritime industry with the means of dealing with very technically advanced vessels of a variety of types. Nevertheless, it will be many years before CFD tools will be able to predict full-scale powering performance without the anchor of model experiments. In the near future CFD tools will become reliably used in hydrodynamic analysis provided a new design strategy is adopted requiring a continuous improvement in the analysis tools and a data base of accumulated engineering experience.

The correlation between CFD, model and full-scale predictions will result in more
efficient hull forms. The vital role which CFD is playing in modern hydrodynamic design and analysis is clearly demonstrated by many applications. The cut-and-try experiments for hull form design will be a thing of the past, and CFD will guide hull form design, changes to the hull form and scaling of the model experiments to full-scale predictions for all aspects of ship performance.

In conclusion, some comments and recommendations are worth to be stated:

- Numerical simulations in close combination with model and full-scale testing, are more and more acknowledged as complementary tools also to provide insight into flow phenomena and to validate numerical codes.
- Combined application of various geometry modelling techniques is essential for an efficient flow of information between computational and experimental models as well as for panelization and grid generation.
- More emphasis should be devoted to understanding discrepancies between numerical and experimental results. Information has to be disseminated as regards capabilities, limitations, and inaccuracies of current CFD tools.
- Integration of CFD into HASD and application of MCDSS will enhance the possibility for fast selection of the best possible designs among feasible alternatives, thus avoiding erroneous decisions in first design stages which imply a difficult correction afterwards.

References


---

**Figure 1**: Shell of HASD procedure