Practical Application of Computer-Aided Hydrodynamic Ship Design

Volker Bertram
Institut für Schiffbau, University Hamburg, Germany
Jochen Laudan
Hamburg Ship Model Basin, Hamburg, Germany

ABSTRACT

Computational fluid dynamics (CFD) can improve hull design for certain ships. The most important CFD methods are panel methods and 'Navier-Stokes' solver. Panel methods compute the inviscid free-surface flow (wave resistance) to evaluate the forebody. 'Navier-Stokes' solver allow evaluation of the aftbody simulating viscous flow usually neglecting free-surface effects. Basic assumptions for both methods are reviewed. Practical applications include a container vessel, a ferry, a SWATH ship, and a tanker.

INTRODUCTION

The development of new ships still is a slow evolutionary process. Series are generally small and time for development of a new ship is generally too short. There is no opportunity for modifications after intensive testing of a prototype, since the prototype is immediately delivered to the customer and has to meet specifications. In addition, modifications will often not be investigated systematically due to limitations in time and budget. Fore- and aftbody, as well as propeller and overall dimensions are modified at the same time so that only the overall effect is known but not the causes in detail. CFD (computational fluid dynamics) methods, which are often cheaper and quicker than experiments and deliver more detailed information, will help the naval industry to overcome this problem in the future. The most important CFD tool in today's practice are panel methods for inviscid flows where free-surface (wave) effects dominate and 'Navier-Stokes' solvers for viscous flows.

INVISCID FREE-SURFACE FLOW

The main difficulty in adapting panel methods developed in the aerospace industry to ship flow problems lay in the free water surface. Here a nonlinear condition has to be fulfilled at an a priori unknown location – at the wave system elevation created by the ship. Furthermore the boundary condition on the ship's hull has also to be fulfilled at an a priori unknown position due to the dynamic trim and sinkage of a ship. Bertram and Laudan [1] review in more detail the historical development of research on this subject. Research still progresses despite available commercial codes based on 'fully nonlinear methods' as problems persist for strong nonlinearities as in ships with strong flare, high-speed applications and sufficient accuracy for quantative prediction.

The wave resistance problem

For the wave resistance problem, water is considered to be incompressible, irrotational and inviscid. Surface tension is neglected. The ship's hull is assumed smooth. Appendages and propeller are neglected. Furthermore, we exclude breaking waves. These assumptions limit us in essence to displacement ships of Froude number $F_n < 0.4$. Conventional cargo ships are not affected by this restriction.

Incompressible potential flow is governed by Laplace's equation for the velocity potential which holds everywhere in the fluid domain. State-of-the-art panel codes fulfill the following boundary conditions to determine the flow field, the wave elevation and the dynamic position of the ship:

1. Water does not penetrate the wetted hull surface
2. Water does not penetrate the water surface
3. Water does not penetrate the sea bottom
4. Water does not penetrate the side walls of a canal
5. At the water surface there is atmospheric pressure
6. At the edge of a (dry) transom stern there is atmospheric pressure
7. Waves created by the ship do not propagate ahead
8. Far away from the ship the disturbance caused by the ship has vanished
9. Waves pass through the boundary of the computational domain
10. The ship is in equilibrium

Bertram and Laudan [1] give some mathematical background on these conditions.

Condition 5 leads to a nonlinear expression in the unknown velocity potential which has to be fulfilled iteratively starting from a linearized approximation. Nonlinear solutions for real ships typically differ by 25% in the wave resistance compared to linear computations, improving accuracy considerably. Although the pressure distribution at the bow is believed to be quite accurate, the wave resistance might still show errors in the order of 50% or more for common discretizations of 400 to 500 panels on the ship hull of a container vessel, unlike test computations for simple geometries such as the parabolic Wigley hull which show excellent agreement with experiments. Reasons for the unsatisfactory accuracy for real ships lie in numerical errors in the approximation of the integration, the viscous interaction in the aftbody and other residual resistance components.

Slow ships such as tankers are especially affected by numerical difficulties. The significant wave length is a quadratic function of speed U. Slow ships create short waves which require in turn fine grids. Storage and CPU requirements increase with $1/U^8$ for small speeds. The wave resistance is numerically very sensitive as pressure integration leads to the subtraction of numbers of same magnitude (the force on the forward half of a ship is typically 10^2 as high as the total force on the ship). As a consequence, grids for tankers are not yet fine enough to give meaningful resistance values. However, the pressure distribution can give valuable insight to improve the hydrodynamic characteristics of the hull.

For very high speeds other problems appear: large areas of breaking waves prevent convergence to nonlinear solutions. This was demonstrated for a SWATH ship, Bertram [2]. For low to moderate Froude numbers the agreement with experiments is still excellent but deteriotes towards large Froude numbers where errors of 200% occur, Fig.1.

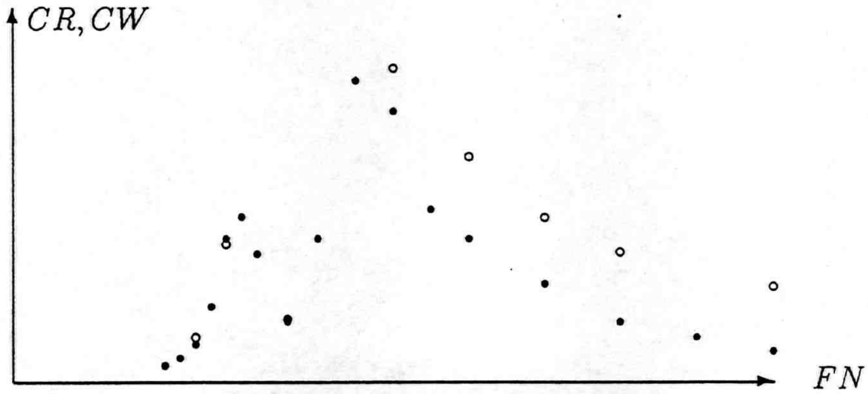

Figure 1: Wave resistance of SWATH ship; ∘ experiment, • computation

Generally, better grids improve accuracy. Trial computations for a modern container vessel with about 400 elements on the hull gave errors in the wave resistance of about 100% for the lowest investigated Froude number which was the most realistic by today's standards, Fig.2. A new grid used about 25% more elements giving a finer resolution mainly on the bow. Furthermore, the new grid was generated automatically from a CAD surface giving a more regular (smoother) distribution of elements. Results were drastically improved.

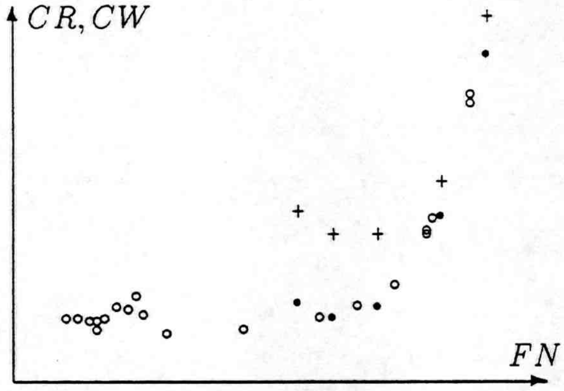

Figure 2: Wave and residual resistance for container vessel
o experiment, + CFD old grid, • CFD new grid

However, despite these shortcomings inviscid CFD is successfully applied to improve hull forms: In a recent project to modify an existing ferry, nine bow forms were investigated. Both length and thickness of the bulb were widely variied. Three forms were selected for further tank tests. The relative performance of the hulls was predicted based on the study of computed wave profiles and pressure distribution on the hulls. Hull 1 was predicted to be slightly better than the original hull, while hulls 2 and 3 promised considerable improvemments. Towing tests confirmed this prediction later: hull 1 gave a power reduction of 2%, hull 2 of 7% and hull 3 of 9% compared to the original hull, Table 1.

Table 1: Power requirements for different bulb forms

Hull	bulb length [m for FP]	power P_D [% of original]	CFD prediction on improvement
Original	3.50	100	
1	3.50	98	slight
2	5.34	93	considerable
3	6.30	91	considerable

VISCOUS FLOW ('NAVIER-STOKES' SOLVER)

The inviscid part of the resistance accounts only for a fraction (< 30%) of the total resistance. Knowledge of the viscous part is of high interest for the evaluation of a ship hull geometry. Unfortunately all existing methods are unable to predict it with adequate accuracy. However, flow phenomena as separation, vortex generation and nonuniformity of the wake field are dominated by viscous effects. Therefore application of viscous flow codes makes sense, as qualitative insight of the flow is possible already today. Viscous investigations of the flow in the aftbody region serve in judging the propulsive properties and are used as input for propeller design.

The Navier-Stokes and the continuity equations are generally considered to be sufficient to describe in principle all real fluid physics for ships. For real ship geometries analytic solutions of this system of nonlinear partial differential equations are impossible. Even if the influence of the free surface is neglected, full numerical solutions are still not possible even on the most powerful computers. For a ship speed of 20 knots the smallest eddies have a length scale of approximately $1\mu m$ and a fluctuation period of $10^{-5}s$. The computational domain covers approximately $10^6 m^3$. To perform a meaningful time-average also over the largest eddies, the integration time has to be approximately $10s$. This discretization of time and space leads to an extremely large number of cells (10^{15} to 10^{20}) which cannot be handled in a reasonable time.

Therefore 'Navier-Stokes' solver split velocities and pressures into a temporal mean and fluctuation part to allow simpler numerical treatment of the equations. The resulting Reynolds-averaged Navier-Stokes equations (RANSE) need for a determinant solution additional equations to describe the turbulence.

Semi-empirical turbulence models supply these additional equations. All known turbulence models are uncertain regarding their applicability for ship flows. Empirical constants were determined for simple flows that did not involve free surfaces or complex geometries at Reynolds numbers comparable to ship problems. The most popular turbulence is the k-ε model. k is the turbulent kinetic energy, ε it's rate of dissipation. The k-ε model can not be applied in the immediate vicinity of a wall. It is therefore always coupled to a logarithmic wall function derived from twodimensional theories. We use the commercial code STAR with the k-ε turbulence model.

For a VLCC project, Bertram et al. [3], viscous flow computations compared two hull versions at model conditions (Reynolds number $Re = 1.17 \cdot 10^7$). The grid had about 130000 cells. A control computation with a finer grid showed no significant differences in the results. Fig.3 shows the pressure distributions in the aftbody region for the original and the modified hull form. The hull modification reduced the low-pressure region at the aft shoulder considerably. This leads to smaller waves and reduced danger of flow separation. Towing tank experiments confirmed the lower resistance of the modified hull. However, the computations did not predict separation which will certainly occur in reality for such a full hull shape. A different turbulence model or treatment of the near-wall region could improve results but would require grid refinements beyond the capability of our current hardware.

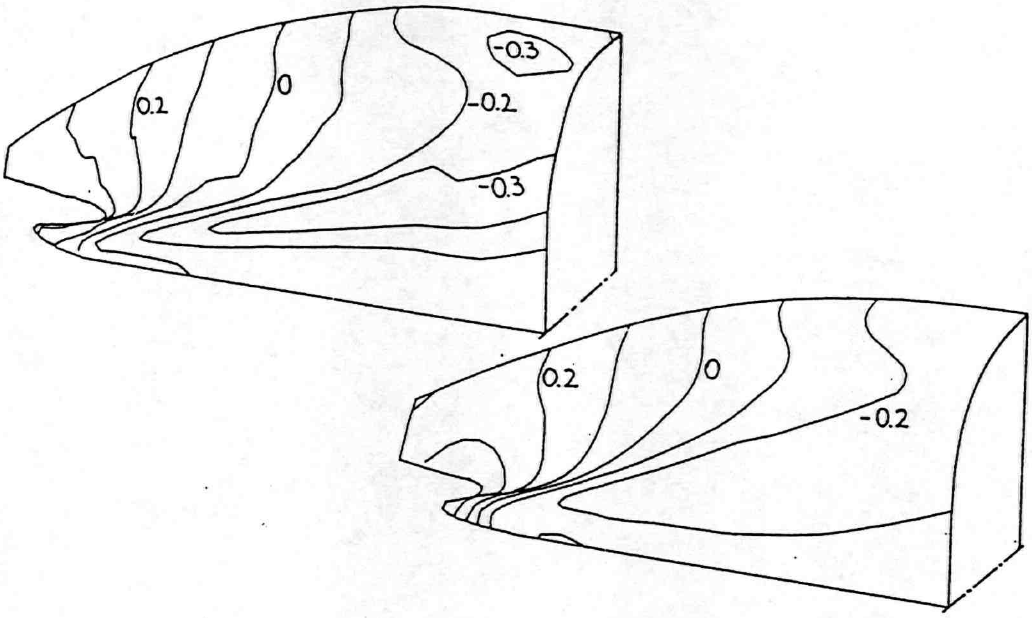

Figure 3: Viscous pressure distribution in the aftbody region of original (left) and modified (right) hull. Isobars spaced by $\Delta C_P = 0.1$. The low-pressure areas have been reduced by the hull modification.

Viscous flow solvers can also be applied to investigate flows about appendages. At HSVA Dr. Streckwall investigated the flow about the aftbody of a twin-screw vessel to see the influence of shaft brackets on the wake in the propeller plane. A profile forming ca. one quarter of a nozzle was designed to improve the flow field. The aftbody including brackets and nozzle was modelled by 200000 elements. The computation showed clearly the influence of the nozzle on the wake, Fig.4., which was not reproduced in wake measurements using pressure probes spaced by the usual 10°. The CFD results stimulated in this case new measurements in a modern cavitation tunnel (HYKAT) with probes spaced by 5°. The finer experimental data reproduced – as expected – the CFD results.

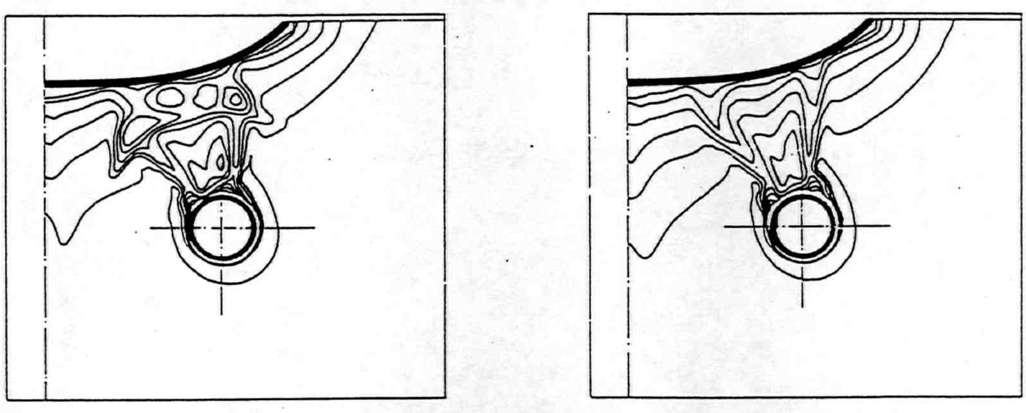

Figure 4: Viscous computations reproduced the wake of a twin-screw ship well. Shown are contour lines of constant longitudinal velocity.
Left:brackets with partial nozzle; Right: brackets without nozzle

PROBLEMS AND PROGRESS

Interpretation of CFD results still poses a major problem for ship designers and CFD experts alike. The absolute accuracy of resistance predictions is still bad and will probably remain so for many years. CFD is used best to compare different variants to select the most promising for model tests. More (documented) experience is needed to design hull forms based on CFD results and maybe to derive correction factors for power prognoses similar to the procedure for towing tank results.

Viscous flow computations still neglect in most cases the waves at the free surface and the propeller-hull interaction. First research applications give rise to hope that these restrictions will be overcome within the next decade. Rapid hardware and software improvement for viscous CFD may also allow better turbulence modelling which is at present possibly the largest source of errors. The solution to the current dilemma with turbulence modelling could be large-eddy simulation (LES). LES simulates directly the large turbulence vortices restricting the uncertain empirical turbulence modelling to the small-scale vortices. Increased direct simulation of turbulence increases accuracy but also CPU time.

The success of introducing CFD methods in ship design not only depends on the quality of the predicted flow quantities but also on economic aspects such as cost and turn around time. In this respect, grid generation is the most critical factor. Considerable progress has been achieved but efforts continue to generate grids almost automatically.

REFERENCES

1. Bertram, V. and Laudan, J. *Computations of viscous and inviscid flow around ship hulls*, Ship Technology Research Vol.40/3, 1993

2. Bertram, V. *SUS-B: A computational fluid dynamics method for SWATH ships*, 2. Int. Conf. on Fast Sea Transportation, FAST'93, Yokohama, 1993

3. Bertram, V., Jensen, G., Laudan, J. and Oltmann, P. *Numerische Schiffsformoptimierung für den E3-Tanker*, Yearbook STG, Springer-Verlag, 1992