

## Resistance and Flow Committee

Committee Chair: Dr. P.S. Jensen

Session Chair: Dr. H. Broberg

### I COMMENTS

#### A comment on CFD uncertainty analysis

E. Campana and A. Di Mascio  
 INSEAN - Istituto Nazionale per Studi ed  
 Esperienze di Architettura Navale  
 Via di Vallerano 139, 00128 Roma, Italy  
 Fax ++ 39-6-5070691

The efforts to rationalize CFD and to quantify the uncertainties of the numerical predictions are surely appraisable and the ASME guidelines are an excellent point of view to define the quality of a computation. Nevertheless, some comments may be done. First of all, in our opinion the assessment of the order of accuracy (guideline 2), the artificial viscosity dependence analysis (guideline 3) and the grid dependence study (guideline 4) are closely related to each other. In fact, the order of accuracy can be evaluated only when the numerical solution lies in the asymptotic range, i.e. when the GCI (grid convergence index) is expected to be small enough. In this condition, the artificial dissipation terms must be negligible with respect to the leading order truncation error (otherwise we are not talking about artificial viscosity). Moreover, when dealing with practical three-dimensional problems, it is extremely unlikely to have the computer resources required to verify the asymptotic range condition.

Then, the GCI is the only index that can quantify the uncertainties in practical calculations, while the order of accuracy assessment should be demanded to some simpler 2D reference cases, for which the number of grid points that can be used is (hopefully) large enough to be confident of the result of the analysis. In this situation, the artificial viscosity should be quantified by

means of a range of values within which the scheme is stable and the order of accuracy does not depend on the particular values of the coefficients.

#### Comment to the Resistance and Flow Committee Report

Prof. Gilbert Dyne  
 SSPA, Sweden

I have a short comment to chapter 2.3.

The Committee seems to give a new definition of the total resistance. I can accept that bare hull resistance, appendage resistance, hull roughness and even environmental effects should be included, but I can not understand how the propeller efficiency and machinery and shaft losses can increase the resistance. Please explain!

#### Comments to the Resistance and Flow Committee Report

M. Ikehata  
 Yokohama National University  
 Yokohama, Japan

1. Does the Committee think whether the solution of Navier-Stokes equation is best among many methods of computational fluid dynamics (CFD), Rankine source method, surface panel methods, vortex lattice method and so on or not?
2. I think that the Reynolds averaged Navier-Stokes (RANS) code is necessary to take off many problems of numerical technics and to improve the methodology in order to make possible flow simulation and resistance estimation at the high Reynolds

number for the full scale ships, that is the goal of CFD. Then CFD will be the strong design tool. How does the Committee think?

3. In Japan Dr. Kodama has made good and great contributions in the field of CFD, such as original grid generation and introduction of the pseudo-compressibility technique. How does the Committee appreciate his contributions?

### CFD Applications to Ship Hydrodynamics (Resistance and Propulsion)

by T. Loukakis, G. Tzabiras and D. Garofallidis  
Laboratory for Ship and Marine Engineering  
Department of Naval Architecture and Marine Engineering, N.T.U.A.

The scope of this contribution is to present additional information concerning CFD applications in viscous ship hydrodynamics, which have been accomplished during the last four years at the Laboratory for Ship and Marine Hydrodynamics of the National Technical University of Athens (NTUA).

A significant part of the research conducted refers to the computation of the resistance and self-propulsion parameters at both model and full scale. Special emphasis has been given to high Reynolds number calculations around full ship forms by solving the complete Reynolds (RANS) equations. A brief description of the examined problems, as published in refs. [1] to [15], is presented in the sequel.

The Reynolds scale effect on the resistance of two characteristic bodies of revolution has been studied in ref. [1]. Extensive numerical tests have been performed to investigate the influence of the body shape, the grid size, the external boundary conditions and the wall function approximation on the calculation of the skin friction and pressure coefficients at model ( $10^6$ ), moderate ( $10^8$ ) and full scale ( $10^9$ ) Reynolds numbers. The effect of laminar regions around the bow on the computed resistance components has also been studied in ref. [2] for three different bodies of revolution at  $Re=10^6$ ,  $10^7$  and  $10^8$ .

Resistance and self-propulsion calculations have been performed for model and full scale for the two tankers which were the test cases of the SSPA-CTH-ITTC-90 workshop. In these computations the double body approximation was adopted, i.e. the free surface was

considered as a plane, while the propeller action was simulated by a simple actuator disk. The required thrust to attain a specified speed has been computed through an iterative procedure and the obtained results have shown that the variation of the resistance and propulsion characteristics depends considerably on the stern geometry. A similar procedure has been employed in ref. [4] to solve an inverse problem, i.e. to calculate the speed of a tanker when the engine characteristics (horsepower and RPM) are known. In this work the wavemaking resistance was computed at various Froude numbers by subtracting the viscous resistance of the double body from towing tank tests at model scale. In ref. [5] a lifting line propeller model was compared to a simple actuator disk approximation when the self-propulsion parameters of a full scale tanker are calculated (double body). A remarkable conclusion of this work was that the two different approaches predicted practically the same horsepower. The sensitivity of the numerical solution of the self-propulsion problem on some crucial actuator disk parameters (circulation distribution, length of the disk, position and diameter of the propeller, influence of propeller torque) has also been investigated for a model and a full scale tanker in ref. [6].

The viscous effects of an additive bulb on the self-propulsion parameters of a full scale tanker have been studied in ref. [7]. Computations around the bulb were carried out employing a C-type mesh, as described in ref. [8]. The results obtained have shown that a considerable decrease of horsepower can be achieved with the bulbous bow, owing to both the skin friction resistance and the effective wake fraction improvements. Ref. [9] deals with the self-propulsion problem for a ship model with asymmetric stern moving at a low Froude number. The most interesting point of this work is that the geometrical asymmetry of the stern interacts with the flow asymmetry caused by the propeller, resulting to a remarkably uniform inflow to the latter. These two references may be considered as examples of CFD applications that highlight the reasons for which geometrical changes improve the hydrodynamic performance of ships.

The problems that a CFD method has to overcome when the self-propulsion of traditional fishing vessels is examined numerically, are presented in ref. [10]. Towing tank experiments have shown extended stern flow separation as well as strong interaction of the propeller with the free-surface, phenomena

that cannot be predicted adequately with conventional CFD tools which, however, may be used effectively to analyse some crucial parameters. Improved near wall turbulence models may also been applied with success to predict separation at low Froude numbers, in contrast to the wall function approach which is not suitable for this kind of flows (ref. [11]).

A more recent and, possibly, more interesting for the ITTC part of the research at NTUA addresses the traditional resistance extrapolation problem. A hybrid method that combines both towing tank experiments and CFD codes has been proposed since 1990 [12]. The concept is to measure the free surface around the model, which is free to obtain its running trim and squat, and then to use it as input to a viscous flow solver in order to calculate the total resistance. The main advantage of this approach is that no other assumption concerning extrapolation is used than that the aforementioned parameters are reasonably similar for the model and the ship. In addition, difficult geometries (e.g. bulbous bows) can be treated effortlessly and the domain which is required to perform the viscous calculations is considerably restricted. Full scale predictions are made assuming that the measured free surface remains unchanged, which is approximately true in the stern region but, however, leads to results which are expected to be on the safe side. Recent detailed studies for a series 60  $c_B=0.6$  model, ref. [13], have produced very encouraging results. The application of the proposed method has given total resistance values that differ by less than 4% than the measured ones at three Froude numbers, i.e. 0.25, 0.316 and 0.35, Figure 1.

The last two references [14] and [15] describe a new numerical method that has been developed to study the steady and unsteady cavitating flow past 2-d hydrofoils. In these works the two-phase flow around the foil is calculated by solving the Navier-Stokes equations considering the fluid as a continuum with variable properties. During the solution, the latter are computed through the existing water-steam formulae, depending on the calculated local flow variables. Due to the significant computing power which is required to solve unsteady cavitation problems, applications of the method have been limited so far to low Reynolds number flows ( $Re=2000$ ).

## References

- [1] G. D. Tzabiras, "A numerical investigation of the Reynolds scale effect on the resistance of bodies of revolution", *Ship Technology Research*, 39, February 1992, pp. 28-44
- [2] G. D. Tzabiras, "A numerical study of laminar flow effects on the resistance of bodies of revolution" Fourth National Conference on Mechanics, E.E.Th.E.M. (HSTAM, IUTAM member), Xanthi, Greece, 1995, pp. 927-937
- [3] G. D. Tzabiras, "Resistance and self propulsion numerical experiments on two tankers at model and full scale", *Ship Technology Research*, 40, February 1993, pp. 20-38
- [4] G. D. Tzabiras, "Calculation of a ship's speed under specified engine characteristics", First International Conference on Marine Technology ODRA-95, Szczecin, Poland, 1995, pp. 49-56
- [5] G. Tzabiras, G. Politis and T. Loukakis "Calculation of propulsion characteristics of a tanker using different propeller models", First International Conference on Marine Technology ODRA-95, Szczecin, Poland, 1995, pp. 65-72
- [6] G. D. Tzabiras, "A numerical study of actuator disk parameters affecting the self-propulsion of a tanker", *International Shipbuilding Progress*, April 1996, pp. 5-47
- [7] G. D. Tzabiras "Numerical prediction of self-propulsion characteristics of full ship forms", STG-Sprechtag : Numerische Schiffshydrodynamik, Potsdam, 1995, pp. 4.1-4.8
- [8] G. D. Tzabiras, "On the calculation of the viscous flow around bulbous or U-shaped bows at zero Froude number", *Ship Technology Research*, 42, February 1995, pp. 31-44
- [9] G. D. Tzabiras "Numerical study of the viscous flow past a ship's model with asymmetric stern", MARIND-96 Conf., Varna, pp. 41-56
- [10] G. D. Tzabiras, A. C. Prifti, G. J. Grigoropoulos and T. A. Loukakis "An advanced CFD method for predicting the propulsive performance of traditional fishing vessels" CADAP-95 RINA Conference, Southampton, 1995, pp 17.1-17.16
- [11] G. D. Tzabiras and A. C. Prifti "Viscous

flow computations past a traditional fishing vessel”, to be presented in 2nd Nat. Conf. on Computational Mechanics 96, Chania, Crete, pp. 163-173

- [12] G. Tzabiras, T. Loukakis and D. Garofallidis, "On the numerical solution of the total ship resistance problem under a predetermined free surface", 18th Symposium on Naval Hydrodynamics, ONR, Ann-Arbor, Michigan, 1990, pp. 1-17
- [13] D. Garofallidis, "Experimental and numerical investigation of the flow around a ship model at various Froude numbers",

Ph. D. Thesis, Dept. of NAME, NTUA, 1996.

- [14] Y. Ventikos and G. Tzabiras, "A numerical study of steady and unsteady cavitation phenomenon around hydrofoils" Proceedings of International Symposium on Cavitation, Deauville, France, 1995, pp. 441-448

- [15] Y. Ventikos, "Numerical investigation of unsteady, cavitating and non-cavitating flows around hydrofoils", Ph. D. Thesis, Dept. of NAME, NTUA, 1996.

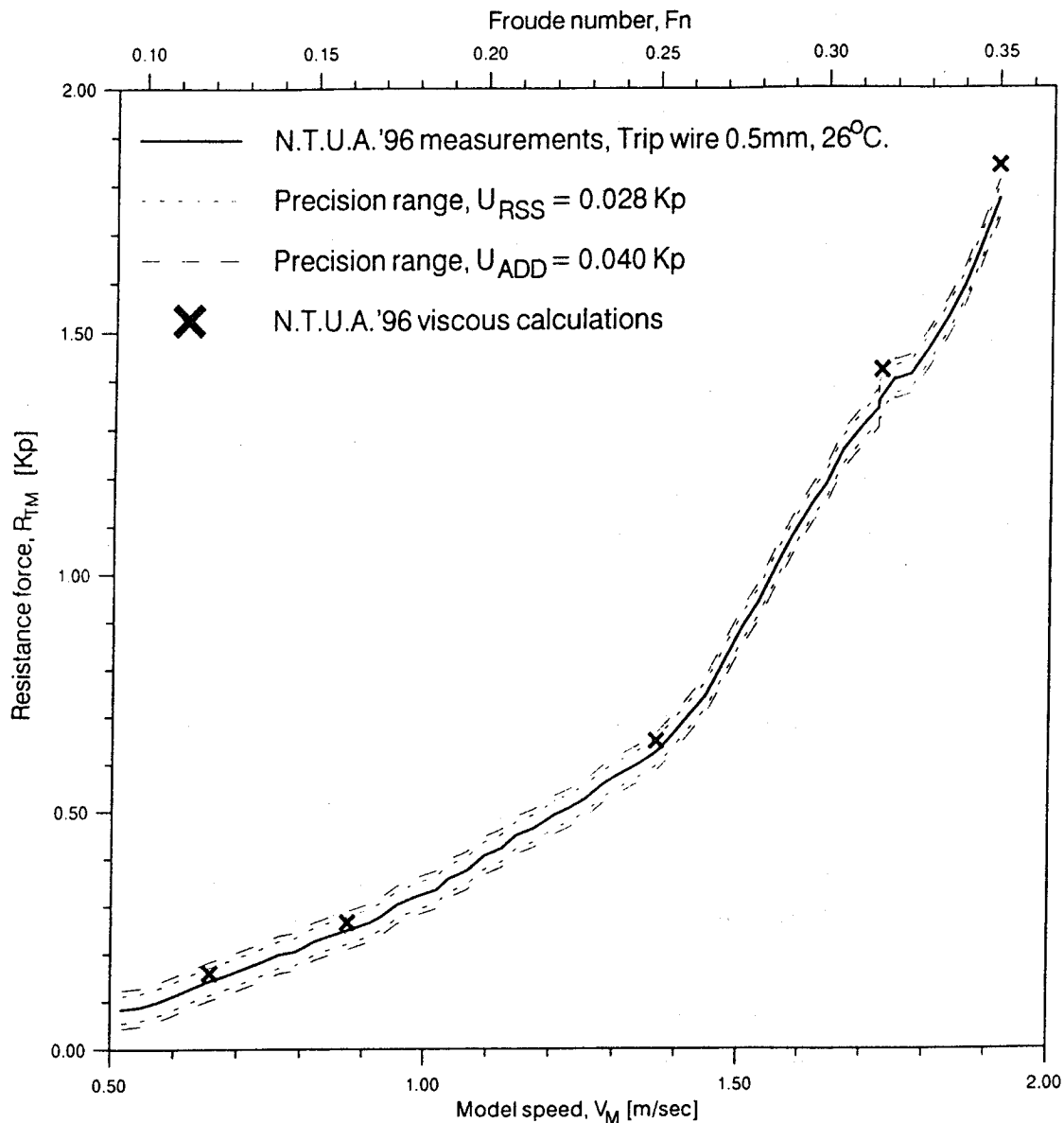


Figure 1 : Measured and calculated total model resistance  $R_{TM}$  with respect to model speed  $V_M$  (series 60  $c_B=0.6$ ,  $L_{BP}=3.048\text{m}$ )

## CFD Approach to Optimizing Design for Ship Hull Lines

by Tieli Li, Yan Lin, Yanying Wang  
Dept. of Naval Architecture  
Dalian University of Technology

CAD methods have been applied for practical initial design for ship lines successfully at DUT in which CFD approach to optimizing design for ship hull lines has been developed. The panel method for inviscid flow in free-surface effect domain and the RANS method for viscous flows in boundary layer and wake domain are applied to CFD system.

### 1. Grid Generation

On the basis of the hull surface accurate

$$\frac{\partial u}{\partial a} + \frac{\partial E}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial E}{\partial \eta} \frac{\partial \eta}{\partial x} + \frac{\partial E}{\partial \zeta} \frac{\partial \zeta}{\partial x} + \frac{\partial F}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial F}{\partial \eta} \frac{\partial \eta}{\partial y} + \frac{\partial F}{\partial \zeta} \frac{\partial \zeta}{\partial y} + \frac{\partial G}{\partial \xi} \frac{\partial \xi}{\partial z} + \frac{\partial G}{\partial \eta} \frac{\partial \eta}{\partial z} + \frac{\partial G}{\partial \zeta} \frac{\partial \zeta}{\partial z} = S \quad (1)$$

where  $u = [\rho, \rho u, \rho v, \rho w]^T$ ,  $E = [\rho u, \rho uu - \mu u_x, \rho uv - \mu v_x, \rho uw - \mu w_x]^T$

$F = [\rho v, \rho vu - \mu u_y, \rho vv - \mu v_y, \rho vw - \mu w_y]^T$ ,  $G = [\rho w, \rho wu - \mu u_z, \rho wv - \mu v_z, \rho ww - \mu w_z]^T$

$S = [0, (\mu u_x)_x + (\mu v_x)_x + (\mu w_x)_x - P_x, (\mu u_y)_y + (\mu v_y)_y + (\mu w_y)_y - P_y, (\mu u_z)_z + (\mu v_z)_z + (\mu w_z)_z - P_z]^T$

$$\begin{aligned} \xi_x &= J^*(y_\eta z_\zeta - y_\zeta z_\eta) & \xi_y &= -J^*(x_\eta z_\zeta - x_\zeta z_\eta) & \xi_z &= J^*(x_\eta y_\zeta - x_\zeta y_\eta) \\ \eta_x &= -J^*(y_\xi z_\zeta - y_\zeta z_\xi) & \eta_y &= J^*(x_\xi z_\zeta - x_\zeta z_\xi) & \eta_z &= -J^*(x_\xi y_\zeta - x_\zeta y_\xi) \\ \zeta_x &= J^*(y_\xi z_\eta - y_\eta z_\xi) & \zeta_y &= -J^*(x_\xi z_\eta - x_\eta z_\xi) & \zeta_z &= J^*(x_\xi y_\eta - x_\eta y_\xi) \end{aligned}$$

$$J = [x_\xi(y_\eta z_\zeta - y_\zeta z_\eta) - x_\eta(y_\xi z_\zeta - y_\zeta z_\xi) + x_\zeta(y_\xi z_\eta - y_\eta z_\xi)]^{-1}$$

The  $k - \varepsilon$  turbulence model has been employed for turbulent flow computation as that

$$(\rho k)_t + \left( \rho u_i k - \frac{\mu_{eff}}{\sigma_k} k_{,i} \right)_{,i} = \rho(G - \varepsilon) \quad (2)$$

$$(\rho \varepsilon)_t + \left( \rho u_i \varepsilon - \frac{\mu_{eff}}{\sigma_\varepsilon} \varepsilon_{,i} \right)_{,i} = \rho \frac{\varepsilon}{k} (C_1 G - C_2 \varepsilon) \quad (3)$$

where the effective viscosity  $\mu_{eff}$  is calculated from  $\mu_{eff} = \mu + \mu_t = \mu + \rho C_\mu k^2 / \varepsilon$  and the turbulent kinetic production term  $G$  is defined as

$$G = C_\mu \frac{k^2}{\varepsilon} \left[ (u_y + v_x)^2 + (v_z + w_y)^2 + (w_x + u_z)^2 + 2(u_x^2 + v_y^2 + w_z^2) \right] \quad (4)$$

in which the turbulence model constants  $C_\mu$ ,  $\sigma_k$ ,  $\sigma_\varepsilon$ ,  $C_1$  and  $C_2$  can be selected from experimental data. In Fig. 2 the vector distribution of transverse velocities for SSPA model 720 is listed at  $x/L = 0.7, 0.8, 0.9$  respectively.

### 3 Calculation of Wave-making Resistance

For an inviscid, incompressible and irrotational (no breaking wave) fluid, the total potential for the wave-making resistance problem can be written by

representation, three dimensional numerical grid around hull surface is generated automatically from the numerical solutions of Poisson-type partial differential equations (PDE) and the density of visula grid can be interacted and controlled by changing control functions. Here, 3D hull surface grid for SSPA model 720<sup>[1]</sup> is given in Fig. 1.

### 2. Calculation of Viscous-Pressure Resistance

Considering the flow around a ship moving steadily ahead, the Navier-Stokes equations and the continuity equation are applied to describe incompressible flow for Newtonian fluid (like water) and can be written in general curvilinear coordinate system  $(\xi, \eta, \zeta)$  as follows:

$$\Phi = Vx + \varphi \quad (5)$$

with corresponding domain and boundary conditions as that

$$\nabla^2 \varphi = 0, \quad \frac{\partial \varphi}{\partial n} = V_n, \quad k_0 \frac{\partial \varphi}{\partial z} + \frac{\partial^2 \varphi}{\partial z^2} = 0 \quad (z = 0), \quad \lim_{z \rightarrow \infty} \frac{\partial \varphi}{\partial z} = 0 \quad (6)$$

The Green function to be fundamental solution of  $\varphi$  may be deduced by Wehausen and Laitone<sup>[2]</sup>

$$G(P, Q) = \frac{1}{r} - \frac{1}{r'} + \frac{k_0}{\pi} \lim_{\mu \rightarrow 0} \int_{-\pi}^{\pi} d\theta \int_0^{\infty} dk \times \frac{\exp\{k[z + \zeta + i(x - \xi)\cos\theta + i(y - \eta)\sin\theta]\}}{k_0 - k \cos^2 \theta - i\mu \cos \theta} \quad (7)$$

in which  $r, r' = [(x - \xi)^2 + (y - \eta)^2 + (z \mp \zeta)^2]^{\frac{1}{2}}$ , and then

$$\varphi(P) = -\frac{1}{4\pi} \iint_{S_n} G(P, Q) \sigma(Q) dS(Q) - \frac{1}{4\pi k_0} \int_{c_n} n_x G(P, Q) \sigma(Q) d\eta \quad (8)$$

Thus, the velocity distribution and the wave-making resistance can be obtained

$$Vn_x = -\frac{1}{2} \sigma - \frac{1}{4\pi} \iint_{S_n} G_n(P, Q) \sigma(Q) dS(Q) - \frac{1}{4\pi k_0} \int_{c_n} n_x G_n(P, Q) \sigma(Q) d\eta \quad (9)$$

$$R_w = -4\pi \rho \iint_{S_n} \sigma(Q) \mu dS(Q) \quad (10)$$

The comparison of coefficients of wave-making resistance both by panel method and by model test for SSPA models No 762, No 720, No 763 and No 764 are given in Fig.3.

#### 4 Design and Optimization of Hull Lines

A curve surface on hull can be described by cubic B spline<sup>[3]</sup>

$$Q_{x,r,z}(U, W) = [U] [V] [P] [V]^T [W]^T \quad (11)$$

Where  $[U] = [U^3, U^2, U, 1]$ ,  $[W] = [W^3, W^2, W, 1]$ , in which  $V$  is base function of cubic B spline and  $P$  is vertex vector of controlled grid and they are

$$[V] = \begin{bmatrix} \frac{1}{6} & \frac{1}{2} & \frac{1}{2} & \frac{1}{6} \\ \frac{1}{2} & -1 & 0 & \frac{1}{2} \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \\ \frac{1}{6} & 0 & 0 & 0 \end{bmatrix}, \quad [P] = \begin{bmatrix} P_{1,J} & P_{1,J+1} & P_{1,J+2} & P_{1,J+3} \\ P_{i+1,J} & P_{i+1,J+1} & P_{i+1,J+2} & P_{i+1,J+3} \\ P_{i+2,J} & P_{i+2,J+1} & P_{i+2,J+2} & P_{i+2,J+3} \\ P_{i+3,J} & P_{i+3,J+1} & P_{i+3,J+2} & P_{i+3,J+3} \end{bmatrix} \quad (12)$$

Hull UV form variation is primary processing for optimizing calculation. Taking  $(x_n, y_n, z_n) = F(x, y, z)$ , where  $(x_n, y_n, z_n)$  is offset of the new form,  $(x, y, z)$  is offset of a selected parent ship and  $F(x, y, z)$  is transformation function for hull UV form variation.

The key of ship UV form variation is to find out a transformed function. The solution of PDE, which are combined with some boundary conditions in three dimensions, can be supposed<sup>[4]</sup>

$$\frac{\partial F(x, y, z)}{\partial x} = BC_x, \quad \frac{\partial F(x, y, z)}{\partial y} = BC_y, \quad \frac{\partial F(x, y, z)}{\partial z} = BC_z \quad (13)$$

where  $BC_x$ ,  $BC_y$ , and  $BC_z$  are geometry boundary conditions of hull.

By using the variation function, a series of hull with different UV form can be produced by a selected parent ship form as listing in Fig. 4.

There are three main elements in the optimizing process. One of them is object function  $F_{obj}$

$$\min(F_{obj}) = [W] \begin{bmatrix} R_w \\ R_p \\ R_f \end{bmatrix} \quad (14)$$

where weight  $W = w_i$ , ( $i = 1, 2, 3$ ), and  $R_w$ ,  $R_p$  and  $R_f$  are wave-making, viscous pressure and friction resistance respectively. The second is constrained functions that may restrain range of primary dimensions,

$$\begin{bmatrix} L_{bp} \\ B \\ T \\ Dispt. \end{bmatrix} \subseteq \begin{bmatrix} (L_{min} & L_{max}) \\ (B_{min} & B_{max}) \\ (T_{min} & T_{max}) \\ (D_{min} & D_{max}) \end{bmatrix} \quad (15)$$

and hull main geometrical coefficients are

$$\begin{bmatrix} C_b \\ C_m \\ C_w \end{bmatrix} \subseteq \begin{bmatrix} (C_{b_{min}} & C_{b_{max}}) \\ (C_{m_{min}} & C_{m_{max}}) \\ (C_{w_{min}} & C_{w_{max}}) \end{bmatrix} \quad (16)$$

and so on. The final is initial parameters which can be got form parent offsets used in beginning optimization.

### 5 Concluding Remarks

- (1) Both computational methods of RANS and Green function to determine viscous pressure and wave-making resistance can be successfully applied to hull form optimization with corresponding CAD system of constructing ship hull lines. This is useful not only to improve ship design but also to decrease the amount of model test.
- (2) It should be pointed that all of numerical methods to predict hull resistance are still not accurate enough and validation from model test and full scale measurement are importance to increase the prediction reliability for ship hull design.

### REFERENCES

- 1 E. Freimanis, Hans Lindgren. Systematic Tests with Ship Models with  $\delta_{pp} = 0.675$ . SSPA, 1957
- 2 Wehausen J V and Laitone E V. Surface Wave in *Encyclopedia of Physics*. Berlin: Springer-Verlag. 1960, 9: 446-814
- 3 Lin Yan, Ji Zhuoshang, Dai Yinsheng. Mathematics Description and Computer Method for B Spline Hull Surface. *Shipbuilding of China*. 1996, (3) (in Chinese)
- 4 Lin Yan, Zhu Zhaohui, Ji Zhuoshang, Dai Yinsheng. Ship Form Description with Function Parameter. *Shipbuilding of China*. 1997, (2) (in Chinese)

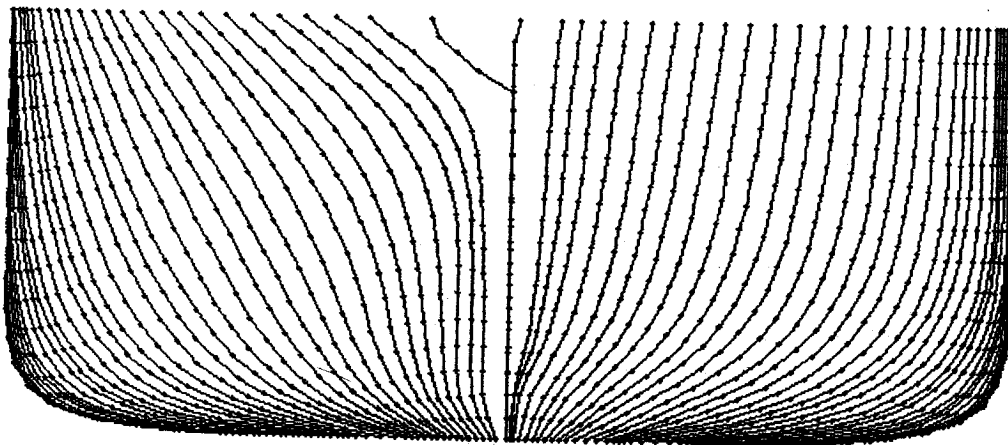


Fig.1 Grid Generation for SSPA Model 720

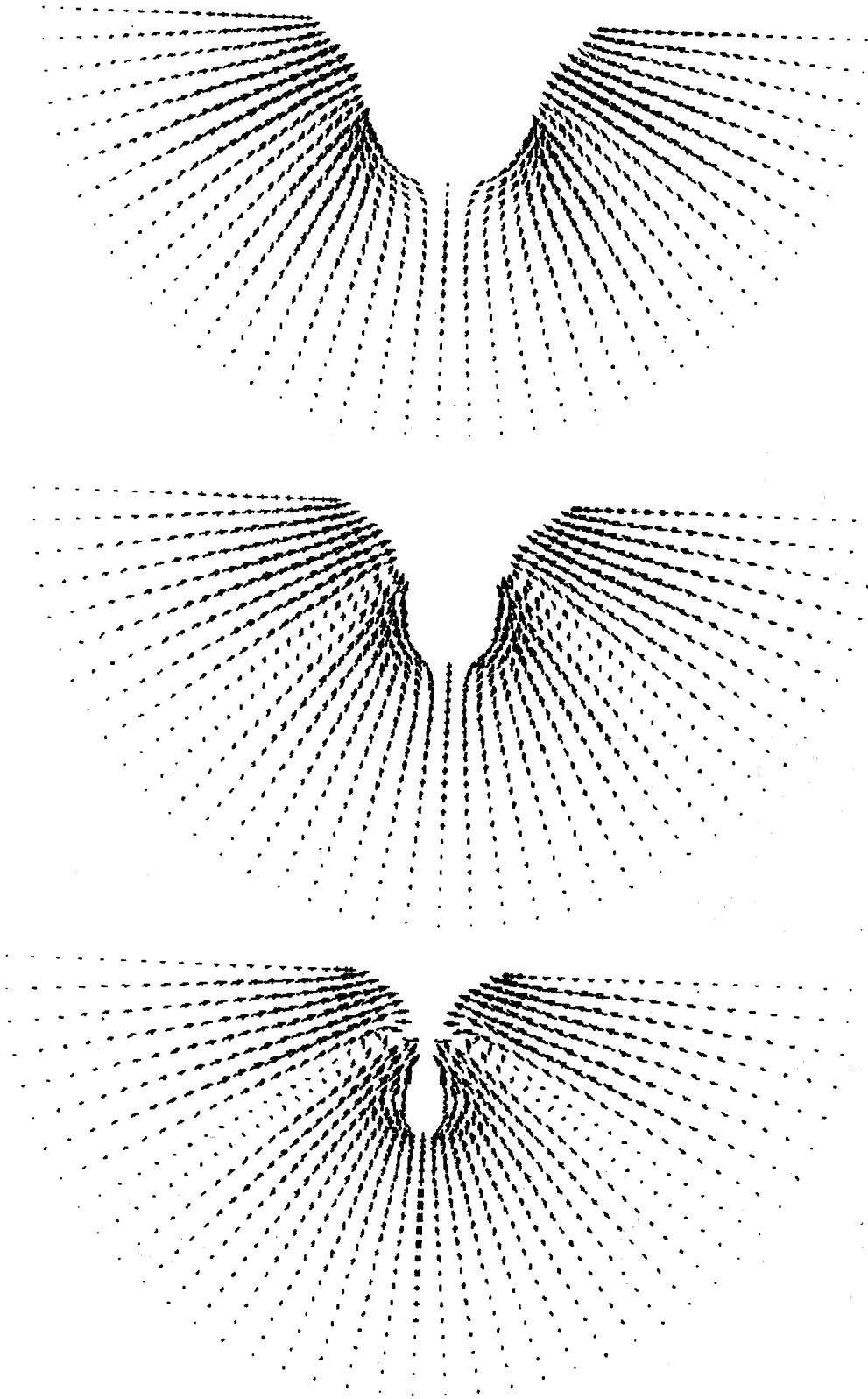


Fig. 2 The Vector Distribution of Transverse Velocities for SSPA Model 720 at  $x/L=0.7, 0.8, 0.9$  Respectively

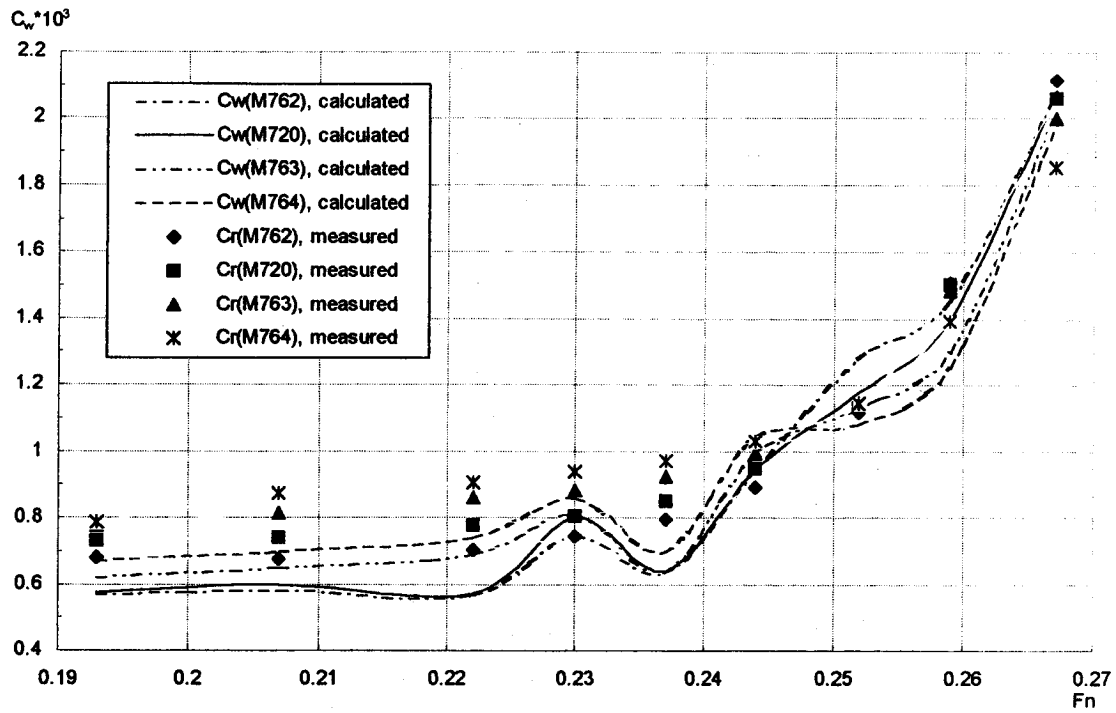


Fig. 3 Comparison of Coefficient of Wavemaking Resistance for SSPA Model 762, 720, 763 and 764

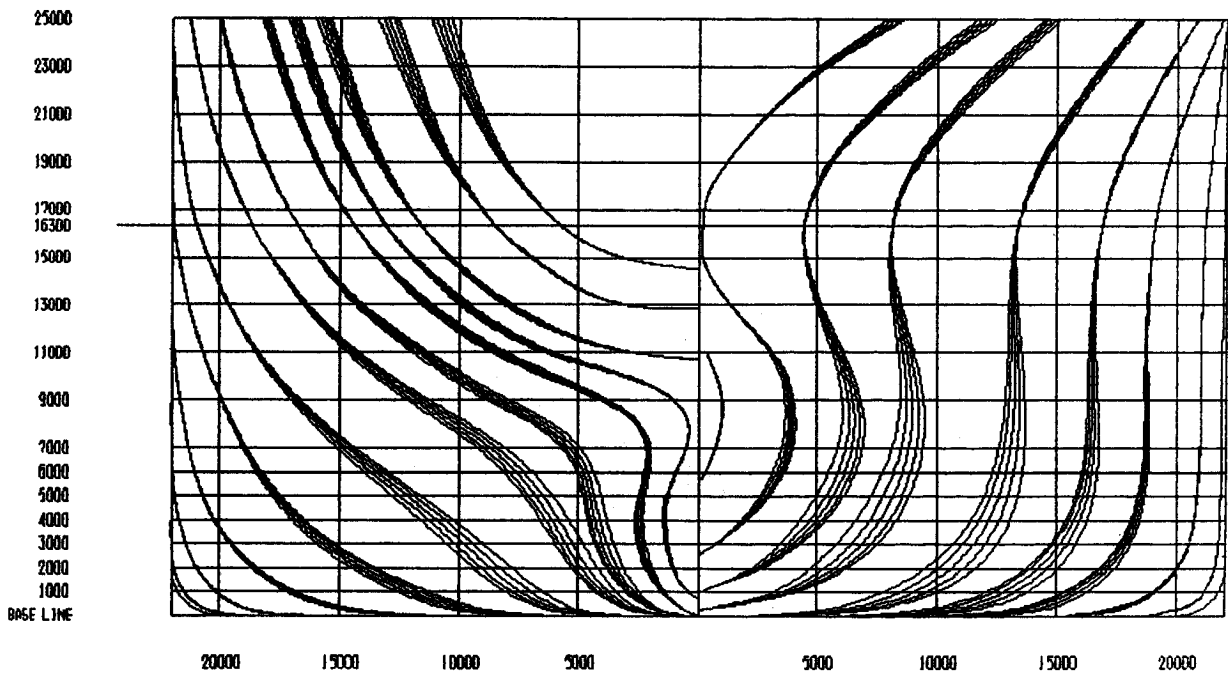


Fig. 4. Hull UV Form Variation for a Bulk Cargo

## II REPLIES

### Response to written discussions - Resistance and Flow Committee

First, we would like to thank both the written and oral discussers of our committee report for their valuable contributions.

The issues raised by E. Campana and A. Di Mascio are well taken, i.e., verification/validation studies for benchmark and simple cases are useful for code development; however, extrapolation procedures to complex cases are unknown. Therefore, uncertainty analysis is, in fact, required for each case.

With regard to the comments of M. Ikehata, we believe that Reynolds-averaged Navier-Stokes (RANS) methods are significantly superior to inviscid-flow methods due to their inclusion of considerably more physics. As reviewed in our committee report, RANS methods are currently useful for quantitative analysis of some hull forms (e.g., medium-speed container ships) and for qualitative analysis of others (e.g., full-form tankers). Furthermore, as also reviewed in our report, rapid progress is being made in RANS development and applicability such that even more complex problems should be amenable to analysis in the near future (e.g., unsteady flows). We regret that we were not able to reference all current work in all areas of ship hydrodynamics and surely recognise the important contributions of Dr. Kodama, but point out that in our report we attempted rather to review the general trends in computational fluid dynamics (CFD) development, including other fields such as mechanical and aerospace engineering. Detailed reviews were given for two areas of ship hydrodynamics, i.e.;

propeller-hull interaction and full-scale Reynolds number.

With regard to the comments of G. Dyne, we agree that consideration of the propeller is very important in ship design and that a conventional propeller is missing in figure 2.1. However, this unintentionally reflects the primary task of our work which is concerned with resistance and flow as opposed to propulsors or propulsion which are the work of other committees. We have not introduced a new definition of total resistance. The propeller efficiency and machinery and shaft losses increase the total power requirement.

With regard to the comments of T. Loukakis, G. Tzabiras, and D. Garofallidis, we appreciate the on-going work on CFD and experimental, ship hydrodynamics at NTUA. As discussed above we were unable to reference all current work in all areas of ship hydrodynamics and apologise for not referencing theirs. Also, as discussed above, the discussers are encouraged to include error bars or confidence limits for both CFD and experimental results. We are curious to know whether the discussers studies of Reynolds number effects for axisymmetric and three-dimensional geometry's confirm the conclusions of our report on form factor dependence on Reynolds number.

Lastly, The comments of T. Li, Y. Lin, and Y. Wang do not concern our report directly such that no specific reply is necessary; however, we appreciate the impressive work presented concerning hull-form optimisation based on a combination of inviscid-flow and RANS methods, including experimental data. As a general recommendation, the discussers are encouraged to include error bars of confidence limits for both CFD and experimental results.