Interaction between Free Surface Flow and Boundary Layer around a Ship Model.

F. VILLEGER,
SIREHNA , 1 rue de la Noë , 44071 NANTES , FRANCE.
B. ALESSANDRINI,
LABORATOIRE DE MECANIQUE DES FLUIDES
ECOLE CENTRALE DE NANTES , 1 rue de la Noë , 44072 NANTES , FRANCE.

Introduction.

We propose here to show that the boundary layer approach is still today an interesting alternative compared to complete Navier-Stokes computation to modelize viscous fluid flow on whole tridimensional bodies, using reasonable amount of CPU time and without restriction on the Reynolds number. Particularly considering an entire body, the boundary layer on the forward part is often too thin to be computed by complete Navier-Stokes equations with appropriate mesh. On the other hand, progress realized these last years allow to compute the boundary layer not only on the forward part but also in many cases on the stern, outside the flow separation region, and this even if the cross flow is reversing (J.COUSTEIX and R.HOUDEVILLE [1]). In addition, taking into account of the free surface increases the set of Navier-Stokes equations, that will need a higher amount of CPU time to be solved. This problem doesn't seem to be completely solved in a near futur. With the boundary layer approach the free surface effect may be taken into account in an easier way through the external velocity field. A similar approach has been recently developped by M.HIKEHATA and Y.TAHARA [2] to study the influence of the boundary layer of the double model flow on the free surface around the Wigley hull.

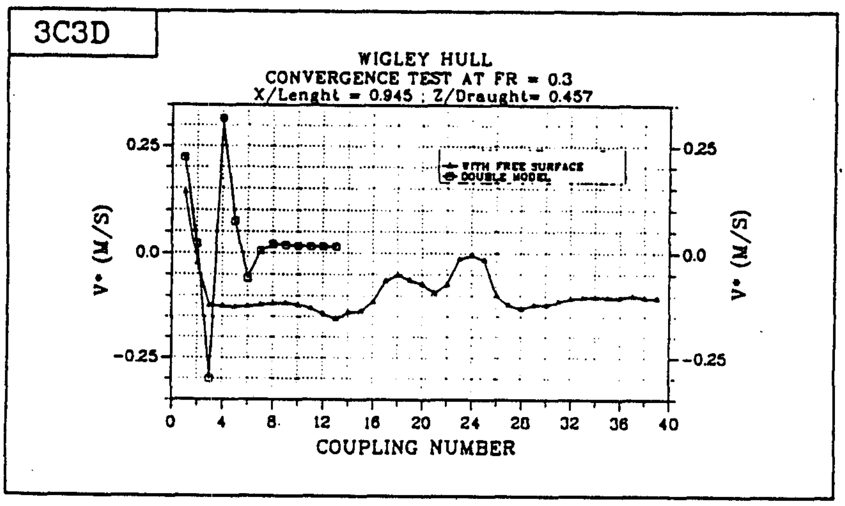
Coupling method.

This paper deals with the interaction between the free surface flow and the boundary layer. A coupling method between boundary layer and inviscid flow is applied to two recent computer codes, a boundary layer program and a wave resistance program. The first one, called 3C3D, solves the three dimensional boundary layer equations along the characteristics lines. It has been developped by the CERT/ONERA and adapted by the authors to the case of a ship model. The second one, called REVA, is a computer code for solving the ship wave resistance problem with linearized free surface condition that has been developped since 1985 at the Laboratoire d'Hydrodynamique Navale of ECN and SIREHNA company in Nantes. This program, based on a Rankine source method provides good results for classical hulls moving at constant forward speed in calm water [3]. The coupling method between both regions of viscid and inviscid flow is based on the transpiration condition on the wall. The boundary condition on the body in the wave resistance program is modified to take into account of the transpiration velocity field calculated by the boundary layer computer code. The convergence test is based on the comparison of the integral quantities field for two successive calculations of the boundary layer.

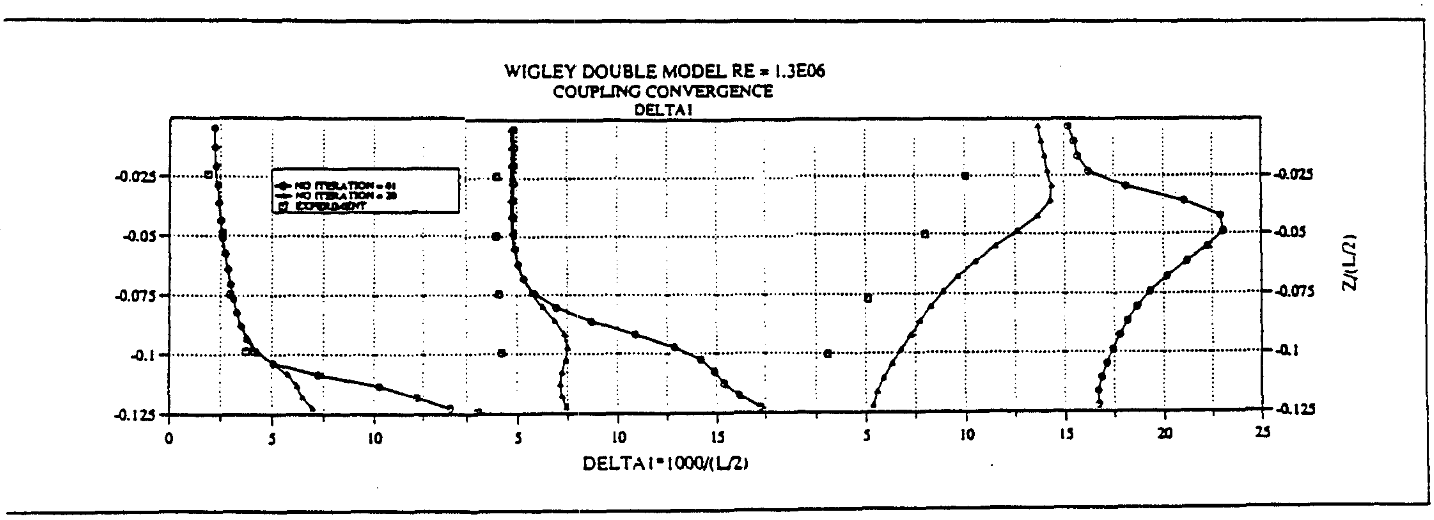
Double model flow.

Calculations are first carried out for double model configuration on the Wigley hull and on the SSPA hull. The calculated boundary layer is beginning at the bow of the ship model and so don't need boundary layer measurement as data to initialize the calculation, unlike most of the classical computer codes. Two types of mesh have been tested for the SSPA hull, a first one leaning on the sections of the ship, a second one based on the transfinite method. The convergence of the coupling method on the Wigley hull at Re=6.7E08 is fast (8 iterations) (fig 1) , but it's more difficult to obtain at low Reynolds number (Re=1.3E06) because of the significant increase of the boundary layer . In this case a solution to update the transpiration velocity is proposed : we examine the convergence in one vertical section at once , progressing gradually from the bow to the stern . We can see the beneficial influence of this method on a stern of Wigley hull by comparison with experiment results (fig 2) .

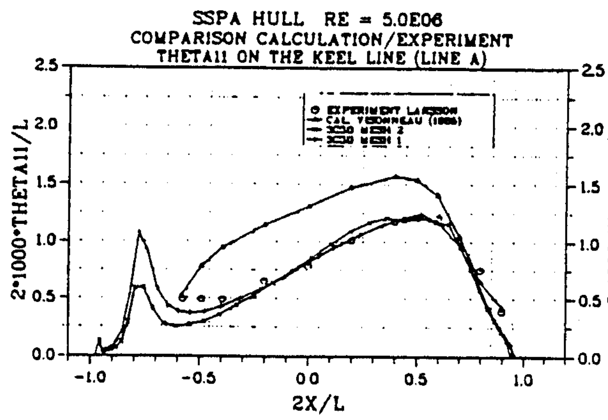
The same difficulty appears on the SSPA hull at proximity of the accumulation line, and the process does'nt converge up to the middle of the boat . Nevertheless , the results obtained at the first iteration , show a better agreement , with LARSSON experiment [4] , than previous computer codes based on the boundary layer approach



- figure 1 -



- figure 2 -



- figure 3 -

Influence of the free surface.

The influence of the free surface on the boundary layer is studied . On the Wigley hull at Re=6.7E08 , the iterative process converges , section by section , for about 40 iterations (fig 1) . On the SSPA hull , results are presented without the iterative process . The influence of the free surface is shown to be significant on the integral quantities (fig 4) and on the wall streamlines (fig 5).

Conclusion.

The interaction between the free surface and the boundary layer has been taken into account through the external inviscid velocity field (potential flow) and coupling method has been realized by use of the transpiration velocity .

The conclusion is that , on the one hand , the results obtained with the coupling process are in better agreement with experiments , and , on the other hand , the influence of free surface on the integral quantities is significant compared to the double model flow .

We can note finally that the convergence is more difficult to obtain with a free surface condition and at low Reynolds number.

The present work was supported by the DRET, French Ministry of Defense, under contract n°90558.

Reference:

[1] J.COUSTEIX, R.HOUDEVILLE, "Singularities in Three-Dimensional Turbulent Boundary-Layer", AIAA Journal, vol.19, No 8.

[2] M.IKEHATA, Y TAHARA, "Influence of Boundary Layer and Wake on Free Surface Flow around a Ship Model", J.S.N.A.Japan, Vol.161, June 1987.

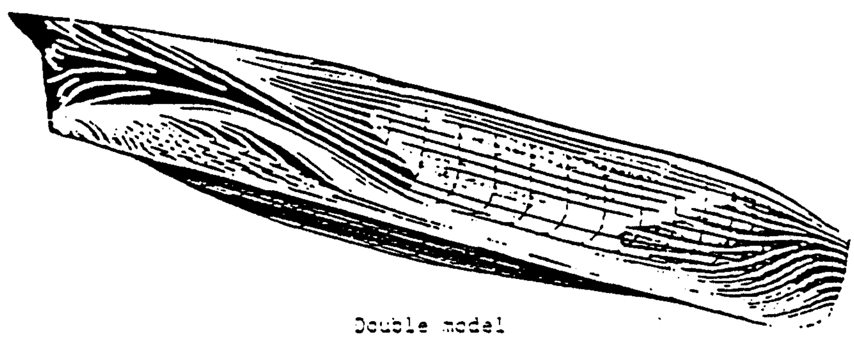
[3] G.DELHOMMEAU, J.J.MAISONNEUVE, "Calculation of Flow Around Several Ship Hulls by Rankine Source Method", International Conference Hydrosoft 86

[4] Proceedings of "International symposium on ship viscous resistance", March 18-20, 1985, Göteborg, SWEDEN.

[5] M.VISONNEAU, "Contribution à l'étude numérique des couches limites tridimensionnelles en mode direct et en mode inverse", thèse, Université de Nantes, 1985.

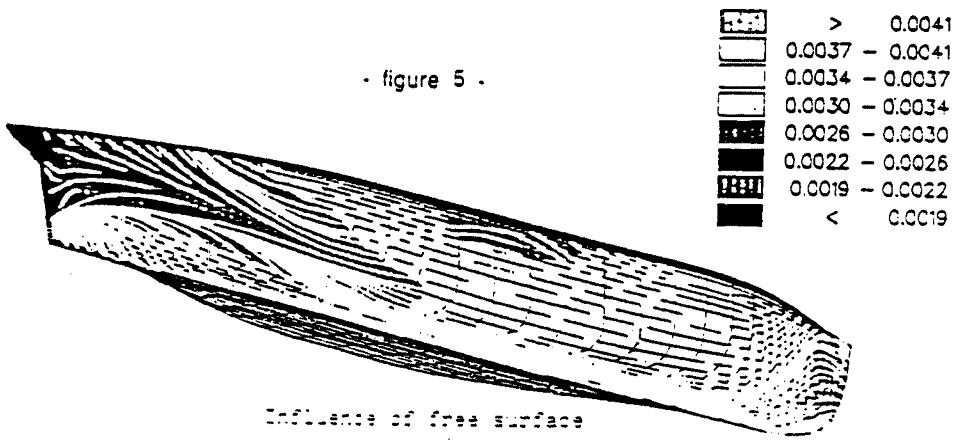
SSPA hull, Re = 5.0E06, Fr = 0,3

Total friction coefficient and wall streamlines :



Double model

- figure 5 -



Influence of free surface
and
corresponding free surface elevation (meters

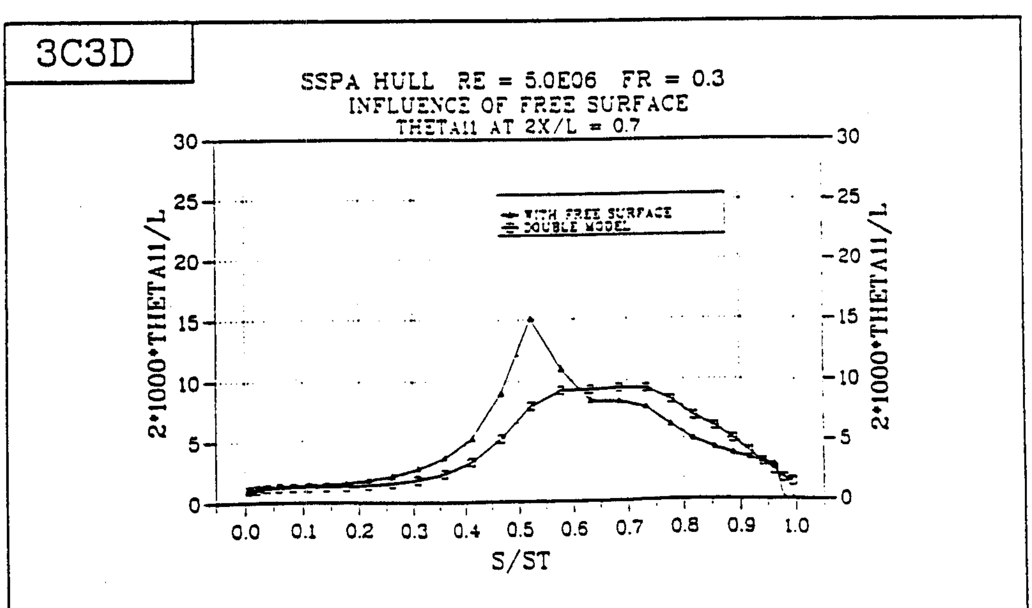


Figure 4

DISCUSSION

RAVEN: When separation occurs, the boundary layer equations become singular if the external velocity is prescribed from an inviscid flow solution. As a result, your coupling procedure will not converge, as you have shown.
An alternative might be to apply a 'strong interaction' coupling, with an inversely posed b.l. problem, at least near the stern. Although probably complicated in 3 D, this might be useful intermediate step before resorting to Navier-Stokes equations. It should allow you to apply b.l. equations to separating flow. Do you consider such an extension?
VILLEGER & ALLESSANDRINI: We agree that when separation occurs, the coupling procedure on the whole body doesn't converge because the boundary layer equations become singular. That is why we propose here to use a coupling procedure not on the whole body but progressing section by section from the bow and stopping just before the separation. This procedure reduced the overpredicted numerical values of boundary layer thickness at the first iteration, particularly near the stern (and before the separation), and gave good results compared with experimental one.
If numerical computation is carried on in the separation domain the boundary layer thickness is oscillating and the procedure will not converge because of the singularity. But we have shown during the session that in some particular case (Wigley hull - Re = 4.5 10^6) the separation on the stern could disappear when such a procedure is used. This is because the separation domain at the first iteration is related to the overpredicted values of boundary layer thickness.
The proposed extension has been considered and seems to be quite complicated for the 3 D case.

LIN: I have just a simple comment. Professor H.C. Chen of Texas A & M University and myself have recently developed a numerical scheme to couple the Navier-Stokes calculation and potential flow calculation to study non-linear free-surface viscous flow around a ship. Good agreement was obtained for series 60 (C_B = 0.6) ship between recent experimental data from Iowa University and our calculations. It took about 30 minutes CPU time on CRAY YMP to obtain the result, we used about 2 000 panels in half of the hull surface and about 60,000 grid cell in the RANS calculation domain. This CPU time requirement is much smaller than the author suggested CPU time requirement for typical Navier-Stokes calculation (≈100 hrs).
VILLEGER & ALLESSANDRINI: The procedure of professors Chen and Lin (Navier-Stokes + Potential flow) seems to be an intermediate step between the boundary layer approach and the full Navier-Stokes approach, for which CPU time is extrapolated here for the same mesh as for the boundary layer and potential flow computation (≈ 10^6 grid cells). This is why the required CPU time mentioned by Professor Lin are intermediate between those proposed during the session.