APPLICATION OF CFD FOR ANALYSIS OF THE SHIP AND PROPELLER FLOW

TOMASZ JAN BUGALSKI AND JAN SZANTYR

Institute of Fluid Flow Machinery
Polish Academy of Sciences
Fiszera 14, 80-952 Gdansk, Poland

Abstract: The paper describes the computer system PANSHIP for analysis of flow around the ship hull moving with constant velocity in calm water, including the effects of free surface and propeller operation. This system calculates the potential flow using the discrete distribution of Rankine sources on the hull. Viscous flow is computed using integral method in the bow section and Reynolds Averaged Navier Stokes equation (RANS) in the stern section of the ship hull. Results of this analysis may be directly used in ship hull design and they may also serve as input for calculation of the unsteady flow phenomena accompanying propeller operation in the non-uniform velocity field generated by the hull. PANSHIP has been verified experimentally and it forms a useful tool available for ship designers and for marine hydrodynamicists.

1. Introduction

The number of ship hydrodynamics problems which can be solved with the help of computational fluid dynamics (CFD) methods is constantly increasing. Practically all research institutions that perform ship model tests employ computer systems for the wave resistance estimation based on the potential theory of the flow around the hull. Many research centres make a great effort in order to build such computer systems that take into account the effects of viscosity and are more accurate and suitable in hull design process — CFD workshop Tokyo [1]. Recently the efforts have also been oriented towards the possibility of the numerical analysis of propulsion characteristics of the ship and propeller.

A computer system for the determination of the potential flow around the hull which takes also into account the influence of the free water surface was developed in Poland in 1994 [2]. The next step of the development of the mentioned system are the procedures that enable the viscous flow calculations. Furthermore, the procedures of numerical simulation of the hull-propeller interaction has been added to the system [3]. The improved version of the system named PANSHIP was verified experimentally. Positive results of the experimental verification made possible the application of the PANSHIP system for the propulsion analysis of the displacement ship.

The non-uniform velocity field in front of the propeller calculated by PANSHIP may serve together with propeller geometry as an input for the analysis of unsteady flow phenomena on the propeller. PANSHIP computer program may reproduce by means of CFD the complete set of model experiments conducted typically in a ship hydrodynamics laboratory for a new ship design.

2. PANSHIP Computer System

The work on 'numerical towing tank' has been carried out for several years. Five years ago a group of four persons was appointed, which has developed algorithms for determination of potential flow around the ship hull including free surface effects, for viscous flow and for propeller–hull interaction effects. These algorithms were the basis for construction of three computer systems: BOS-I funded and developed by Ship Design and Reseach Centre (CTO) for initial resistance analysis of ships [4], PANSHIP developed at IFFM and optimised for education and reseach on hull–propeller interaction, and CISAKU sponsored by the Commitee for Scientific Reseach (KBN) specialised for computation of acoustic pressure field generated by a moving ship with operating propeller [5].

2.1 PANSHIP algorithm

The process of solution starts from the analysis of flow around a "double" hull, assuming that the free water surface is a rigid flat wall.

The numerical solution starts with discretisation of the ship hull wetted surface, which is divided into a grid of flat, quadrilateral elements.

The free water surface is divided into a number of elements, in the manner similar to the hull surface.

The system of ship generated waves is obtained through a transformation of the Bernoulli waves. This transformation is based on an original solution of the Green function for a single moving pressure pulse [6], and equally original method of superposition of such solutions, leading to a very short computation time.

Then, the streamlines on hull surface for potential flow are determined while the influence of free surface is taken into consideration. The parameters of boundary layer located on the hull surface are calculated along those streamlines with the help of integral methods (Truckenbrodt method among others). Those parameters enable the determination of velocity field inside the boundary layer and its thickness distribution.

The dimensions of the hull are increased by displacement layer thickness and for such calculated shape of hull the flow around is being determined again. If calculations are coupled with consideration of operating screw propeller, then the velocity field on the hull surface is appropriately modified.

The final determination of flow parameters is conducted through an iterative process in which the equivalence of propeller thrust and hull resistance is the convergence condition.

It should be mentioned that for better estimation of velocity field in propeller neighbourhood the additional calculations behind the stern with the use of RANS procedure are carried out.

The propeller influence on the hull flow, including the free surface effect, is taken into account by superimposing the propeller-induced velocity field on the hull flow calculated without propeller. The propeller-induced velocity field enters into the kinematic boundary condition, changing the solution of the potential flow around the hull and influencing the viscous flow solution. As the propeller-hull interaction is mutual, an iterative process must be introduced. The starting point of this process is the mean inflow velocity to the propeller estimated from approximate empirical relations. Together with initial assumption about propeller rate of rotation it enables calculation of propeller loading and estimation of the induced velocity field. This field is then introduced into the kinematic boundary condition on the hull and it is later overlaid with the resulting hull-induced velocity field, leading to the new estimation of the hull pressure distribution and hull resistance, which now includes the propeller suction force. In the next iteration the new, analytically established, inflow velocity field to the propeller is employed for estimation of propeller loading, leading to the new, corrected values of the propeller-induced velocities. In the following iteration step the rate of propeller rotation is corrected on the basis of comparison of the hull resistance (including suction) with the propeller thrust. This sequence of corrections (first step-velocity, second step-rate of rotation) leads to the ship or model self-propulsion point (depending on the scale used in calculations).

The propeller-induced velocity field is calculated in a special subprogram based on the lifting surface theory. This theory employs discrete distribution of vortices for modelling propeller loading and discrete distribution of sources/sinks for modelling of propeller blade thickness. The method enables calculation of induced velocity field for single or twin screw configurations, with up to seven blades per propeller.

2.2 Numerical implementation

The PANSHIP system (Figure 1) is composed of a large number of subroutines organised into ten calculation modules, one data input module and five modules for graphical presentation of the results. The structure of the system enables using all subroutines almost independently. All elements of the system are written in FORTRAN. Calculation may be performed on Silicon Graphics workstations or IBM compatible PC's. Currently used SGI INDY R5000SC 150MHz/128MB RAM computers allow for the maximum of 20000 panels on the hull wetted surface (250 elements along X axis with 80 elements along each frame). The region of the free water surface extending 0.5 Lpp forward of the bow, 1.0 Lpp aft of the stern and 3/8 Lpp to both sides from the plane of symmetry is divided into 4200 panels (140 elements along X axis with 30 elements in the transverse direction).

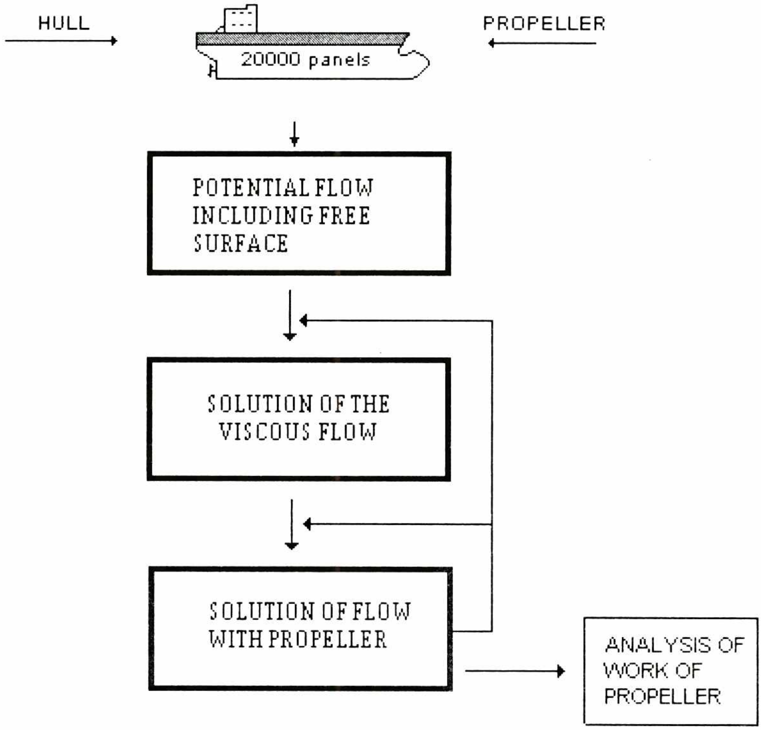

Figure 1. *Block diagram of PANSHIP computer system*

2.3 Results and their application

The results of calculation performed with the use of PANSHIP system contain the following information:

static pressure distribution on the hull without propeller (Figure 2), static pressure

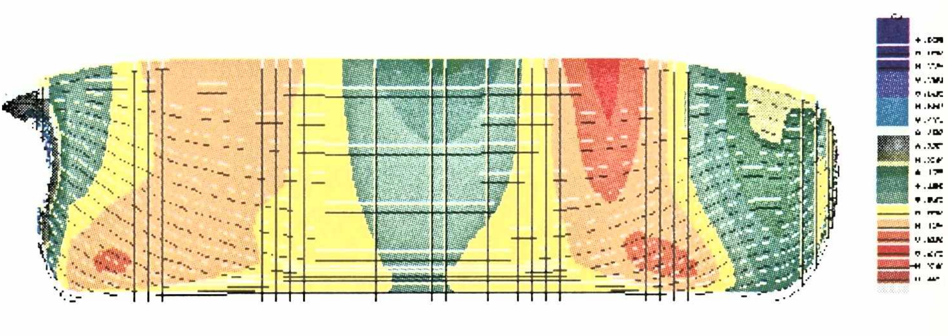

Figure 2. *Static pressure distribution on the hull of ship without propeller*

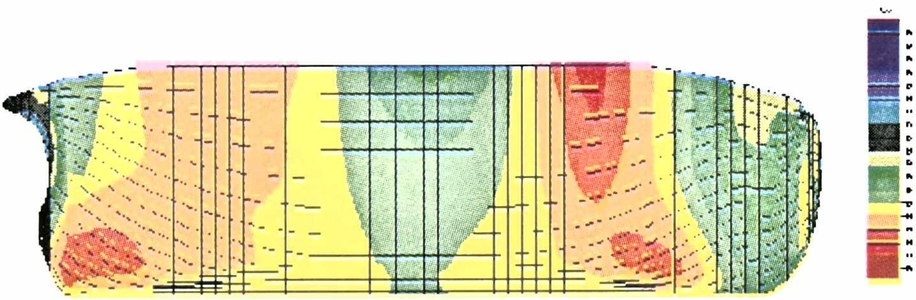

Figure 3. *Static pressure distribution on the hull of ship with operating propeller*

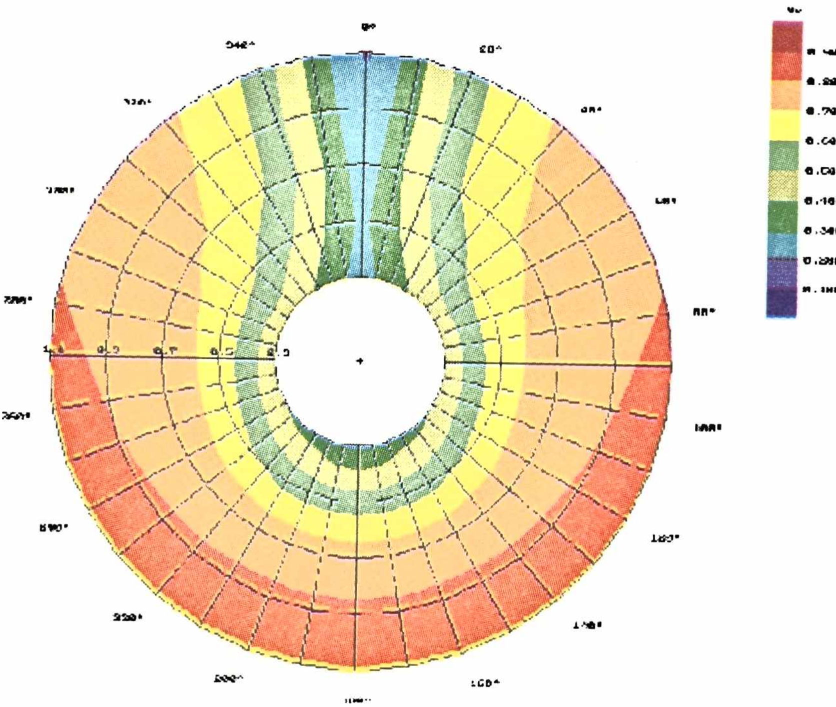

Figure 4. *Nominal wake field at propeller plane*

distribution on the hull with operating propeller (Figure 3), nominal wake field at propeller plane (Figure 4), effective wake field at propeller plane (Figure 5), wave system on the free surface (Figure 6), wave profile along the waterline, streamlines on the hull surface, value of the wave resistance, viscous resistance, thrust of propeller and thrust deduction factor. These results are helpful in an introductory analysis of ship hydrodynamic properties.

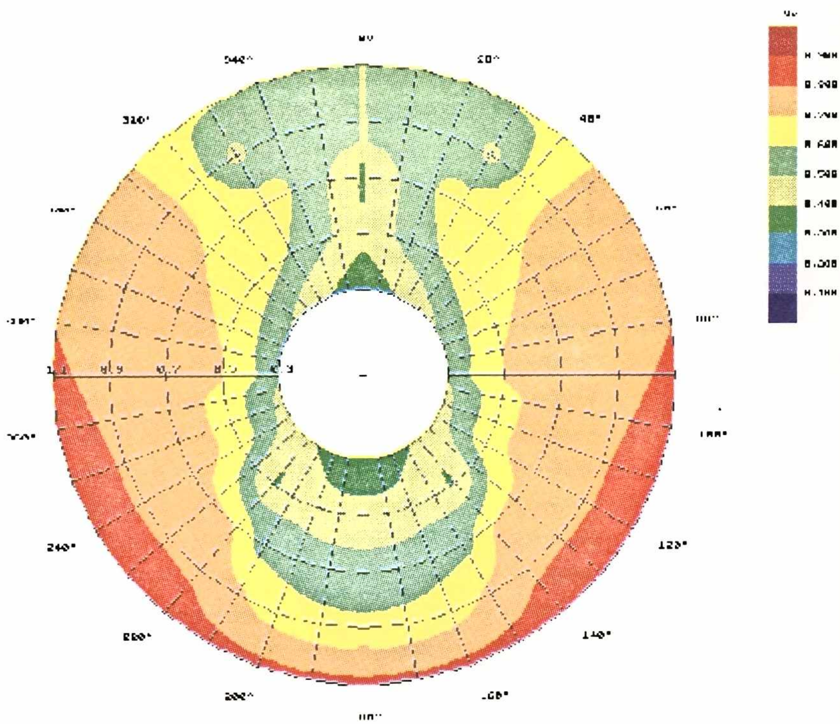

Figure 5. *Effective wake field at propeller plane*

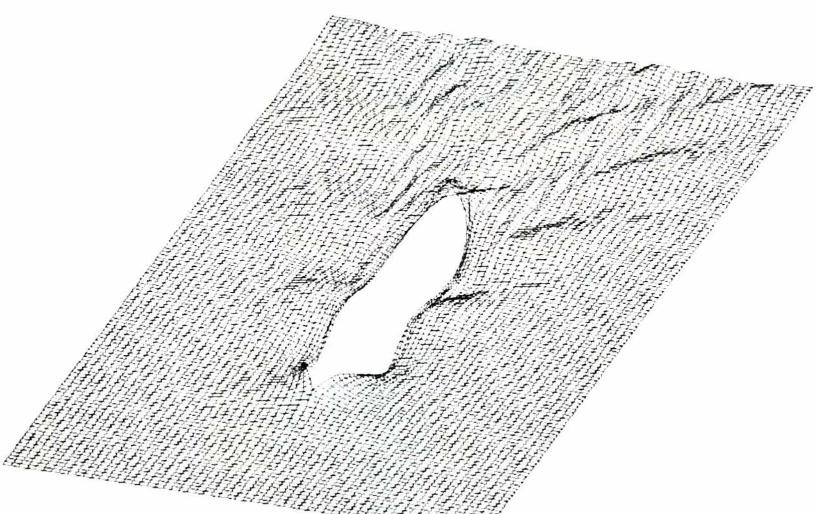

Figure 6. *Perspective view on wave pattern generated by the moving ship on the free surface*

The non-uniform velocity field in front of the propeller, calculated by means of PANSHIP may be employed as input for the analysis of unsteady flow phenomena on the propeller, including different forms of cavitation, variable hydrodynamic forces and pressure pulses. A method based on surface panel representation of the propeller has been developed at IFFM for this purpose [7].

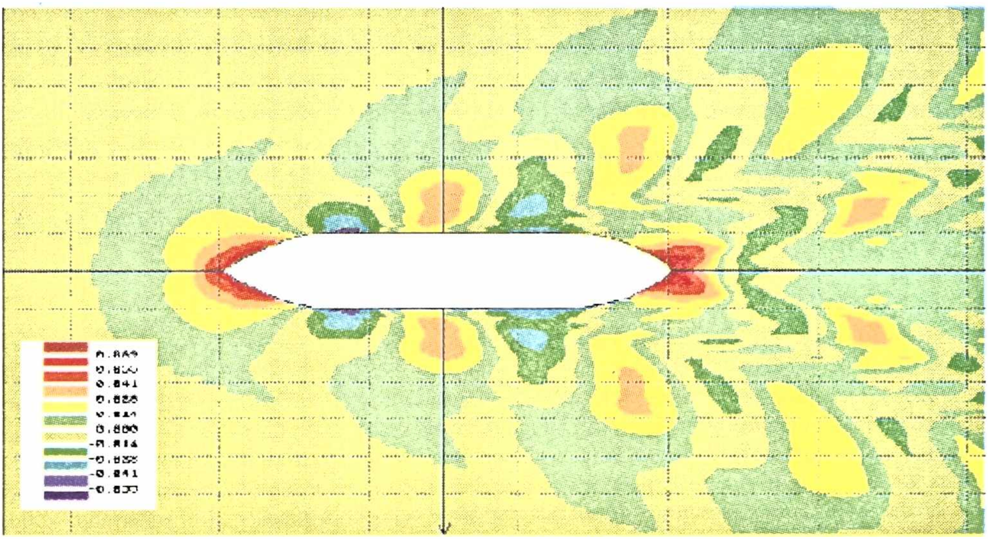

Figure 7. *Wave system about ship hull-lines of equal wave elevation*

3. Conclusion

A CFD computer system has been presented in this paper, which enables a complete analysis of ship and propeller flow phenomena. The scope of this analysis corresponds to the extent of model experiments conducted in the ship hydrodynamics laboratory during development of a new ship design. Practical application of the described system may limit costly and time-consuming experiments to the final verification of the design. Results of calculations of PANSHIP system are continuously being confronted with experimental data, confirming the correctness of basic theory and of numerical procedures.

References

[1] *Proceedings of CFD WORKSHOP TOKYO*, **vol. 1 and 2**, (1994): An International Workshop for Improvement of Hull Form Design, Tokyo

[2] Bugalski T., Koronowicz T., Szantyr J., Waberska G., **vol. 1**, (1994), pp. 5-12, *Computer System for Calculation of Flow, Resistance and Propulsion of a Ship at the Design Stage*, 5th CADMO'94, Southampton

[3] Bugalski T., **vol. 1**, (1997), pp. 2.3-2.8, *Modern Method for Investigation of Hull-Propeller Interaction Phenomena using CFD*, International Conference NAV97, Sorrento

[4] Bugalski T., Bech T., **vol. 1**, (1995), pp. 39-46, *BOS-L a Computer System for the Ship Flow Investigation*, International Conference on Problems of Marine Propulsion, Gdansk

[5] Koronowicz T., Szantyr J., Bugalski T., **vol. 4**, (1997), pp. 2-8, *Theoretical Model for Determining the Pressure Field resulting from Hull Flow and Operation of Marine Propeller*, Polish Maritime Research No. 3

[6] Koronowicz T., **vol. 427/1376/94**, (1994). *Numerical Solution of the Formula Describing the Waves Generated by a Pressure Impulse on the Free Surface of Water*, IFFM Report, Gdansk (in polish)

[7] Szantyr J., **vol. 41**, (1994), pp. 223-242, *A Method for Analysis of Cavitating Marine Propellers in Non-uniform Flow*, Intern. Shipbuilding Progress, No. 427