POLISH MARITIME RESEARCH 4(62) 2009 Vol 16; pp. 3-8
10.2478/v10012-008-0049-x

Numerical analysis of influence of ship hull form modification on ship resistance and propulsion characteristics

Part I

Influence of hull form modification on ship resistance characteristics

Tadeusz Szelangiewicz, Prof.
Tomasz Abramowski, Ph. D.
West Pomeranian University of Technology, Szczecin

ABSTRACT

After signing ship building contract shipyard's design office orders performance of ship resistance and propulsion model tests aimed at, apart from resistance measurements, also determination of ship speed, propeller rotational speed and propulsion engine power for the designed ship, as well as improvement of its hull form, if necessary. Range of ship hull modifications is practically very limited due to cost and time reasons. Hence numerical methods, mainly CFD ones are more and more often used for such tests. In this paper consisted of 3 parts, are presented results of numerical calculations of hull resistance, wake and efficiency of propeller operating in non-homogenous velocity field, performed for research on 18 hull versions of B573 ship designed and built by Szczecin Nowa Shipyard.

Keywords: ship hull geometry; computational fluid dynamics (CFD); resistance; wake; propeller efficiency

INTRODUCTION

The whole ship design process consisted of 6 ÷ 7 stages can be split into two parts: pre-contractual design, i.e. before signing contract, and post-contractual design. In the design process the designer has to find an optimum design of a given ship, which fulfils shipowner's requirements as well as technical conditions resulting from rules of ship classification institutions and/or organizations dealing with safety at sea (e.g. IMO). However crucial decisions are made already during pre-contactual design phase in the situation where very scarce information is known about designed ship. The basic and simultaneously important parameters determined by shipowner, which have decisive impact on future operational profitability of the ship are: ship speed, propeller rotational speed and propulsion engine power at which the ship speed could be maintained. Overall propulsion efficiency which should be obviously as high as possible, is associated with ship speed and propulsion engine power.

The shipowner obviously expects that the designed ship will reach its assumed service speed at the highest value of propulsion efficiency, i.e. at the possible lowest power delivered to propeller, that also leads to the possible lowest hourly or daily fuel consumption.

Already in the preliminary design stage the ship designer, basing on approximate methods, e.g. [4, 5], determines ship resistance, propulsion power, propeller parameters and overall propulsion efficiency.

After signing the contract further improving the design with respect to available speed and propulsion efficiency takes place mainly on the basis of appropriate resistance and propulsion model tests of the ship in question. During model tests in an experimental tank, apart from resistance measurements, nominal wake and propeller design, also propulsion engine power, propeller rotational speed, and necessary modifications of ship hull form, mainly of its stern part, aimed at obtaining the maximum available ship propulsion efficiency, are determined. However number and range of ship hull modifications during model tests is limited because of high cost and time consumption of such tests. Hence it is not certain whether the elaborated ship hull form together with the designed propeller will ensure the lowest propulsion power, simultaneously at the highest propulsion efficiency, for the assumed speed.

Such possibilities are offered by numerical methods especially those of the computational fluid dynamics (CFD) [1, 2, 3, 6, 8, 9]. The methods can be successfully applied just before contract signing that should contribute to improving designed ship's quality as well as to rising competitiveness of a shipyard which applies such methods. This paper presents results of numerical analyses of influence of ship hull form modifications on resistance characteristics, wake and propeller efficiency, performed for a ship designed and built by Szczecin Nowa Shipyard. Hull form of the built ship is assumed initial one for process of its successive modifications, and to verify the numerical calculations available results of basin model tests and sea trials are used.

COMPUTER SYSTEM APPLIED TO NUMERICAL ANALYSES

Research on flow around floating objects as well as modelling process of their form can be performed by means of various computer programs making use of CFD methods. Commercial packages of computational programs such as CFX, Fluent, Commet, V-Shallo, Marine or the programs elaborated and implemented by research centres, such as PANSHIP, BOS, HPSDK, are in wide use. Prior to actual calculations, hull form of an investigated ship is prepared by using e.g. ACAD or MaxSurf software, and next computational grid is prepared by means of such programs as Gridgen, ICEM, or Gambit.

On the basis of analyzed possibilities of available computer programs, FLUENT system was chosen to investigate viscous flow around ship hull, propeller and rudder, and GAMBIT system was applied to build numerical grids. Both the systems (multi-processor licence) in 64-bit version was installed under control of LINUX operational system.

The selection of FLUENT system has been mainly justified by its wide computational capability for complex systems such as ship hull, rotating propeller, blade rudder (possible application of sliding computational grids), as well as by possible performance of simulation calculations for non-stationary states (e.g. a ship moving with non-stationary speed).

Preparatory calculations were performed by using PC computers, and actual numerical calculations by means of NEPTUN computer cluster installed in the Department of Ocean Engineering and Design of Marine Systems, Faculty of Maritime Technology, West-Pomeranian University of Technology, Szczecin. The NEPTUN cluster is built of Xeon 4-core processors and its capacity is of about 1T floops. Such computer system makes it possible to shorten calculation time and to conduct calculations for modelled ship hull also in natural scale.

In the frame of realization of the research project in question a few tests and experiments were performed in advance to determine most optimum configuration of the cluster with a view of effectiveness of calculations conducted by means of the multi-processor version of FLUENT system.

For the processing of calculation results in graphical form a graphical work station fitted with Xeon 4-core processors, PNY 3500 professional graphical card, as well as Raid disk matrices (archiving the data and calculation results), was used.

NUMERICAL CALCULATIONS OF B 573 SHIP'S HULL MODEL

On the basis of an agreement signed between the head of Ship Design Office of Szczecin Nowa Shipyard and the head of the Department, numerical analyses in question were performed for a B 573 ship (Tab. 1) for which almost full scope of design documentation including results of basin model tests and sea trials of the built ship, was available.

The ship's hull body lines are shown in Fig. 1 – this is the initial hull form taken for further modification of its geometry.

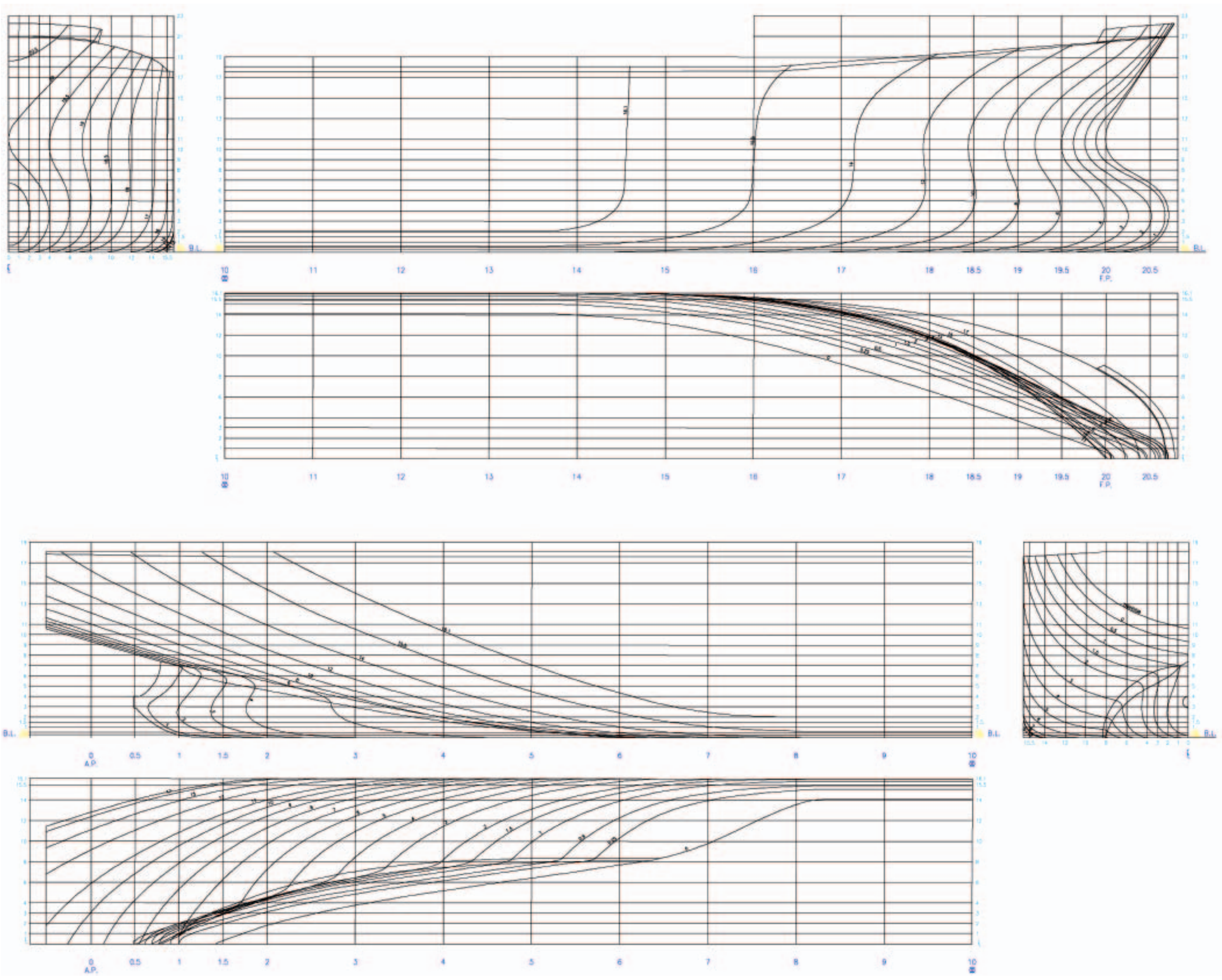

Fig. 1. Body lines of the analyzed hull form of B 573 ship

Tab. 1. Main parameters of B 573 ship in various loading conditions

Length between perpendiculars	L_{PP}	172.00 m	172.00 m	172.00 m
Waterline length	L_{WL}	176.17 m	176.41 m	176.52 m
Hull breadth	B	32.20 m	32.20 m	32.20 m
Draught	T	10.50 m	11.30 m	12.00 m
Volumetric displacement	∇	46540 m^3	50500 m^3	54010 m^3
Wetted surface area	S	7874 m^2	8201 m^2	8477 m^2
Block coefficient	C_B	0.800	0.807	0.813

Prior to actual numerical analyzes a test which consisted in comparing results of resistance calculations made for the initial hull form (Fig. 1) by using CFD method with results of basin model tests [7], was performed. The calculations were conducted for the ship in 1:25 model scale. They were performed by modelling the flow around with the use of RANS equations where turbulence was modeled by means of Reynolds stresses (RSM). Boundary layer was modelled by means of standard functions approximating the flow, with the parameter $y^+ = 50$.

To discretize the domain a non-structural numerical grid (Fig. 2) with prismatic elements in boundary layer was elaborated, which was then converted to that of polyhedral elements (Fig. 3). The non-structural grid was elaborated by using Gambit system and the conversion - by Fluent system.

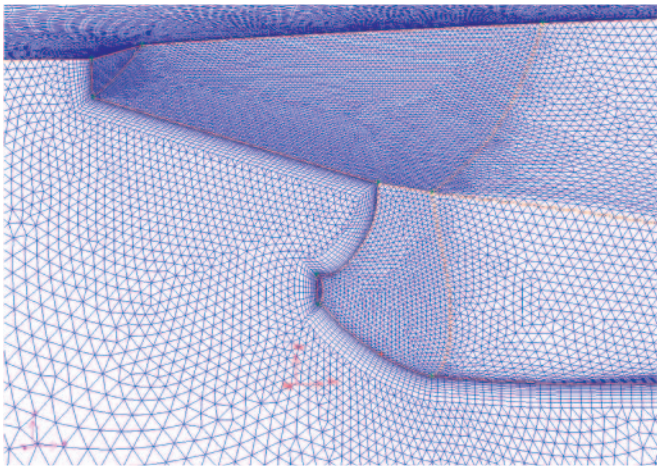

Fig. 2. Non-structured tetrahedral grid

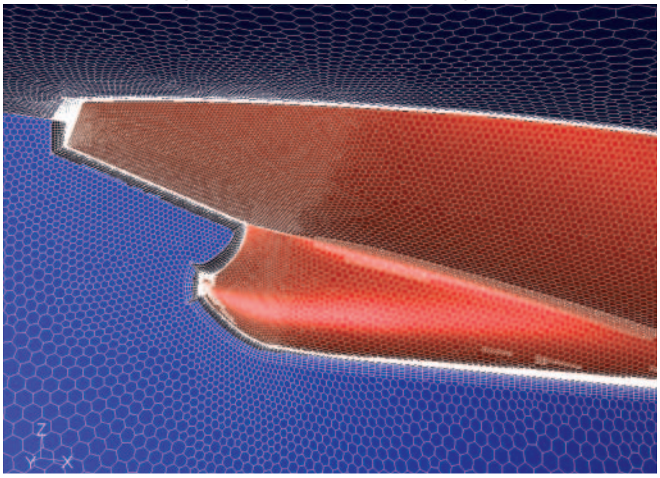

Fig. 3. Polyhedral grid obtained from conversion of non-structured tetrahedral one

Conversion of non-structural grids to polyhedral ones makes it possible to decrease number of grid elements by 3 to 5 times and to improve their parameters, especially the element skewness parameter. The whole number of elements in all variants of calculations for particular draughts was finally contained in the range from 650 to 750 thousand.

In all calculation variants convergence of iteration process was controlled both for residual values of all equations and coefficients of forces generated on the hull. An example convergence run (which was similar for all the tests) is presented in Fig. 4. Iteration tests was terminated when all the remainders (residuals) were below the value of $1e^{-04}$ and resistance coefficient value did not undergo changes any further.

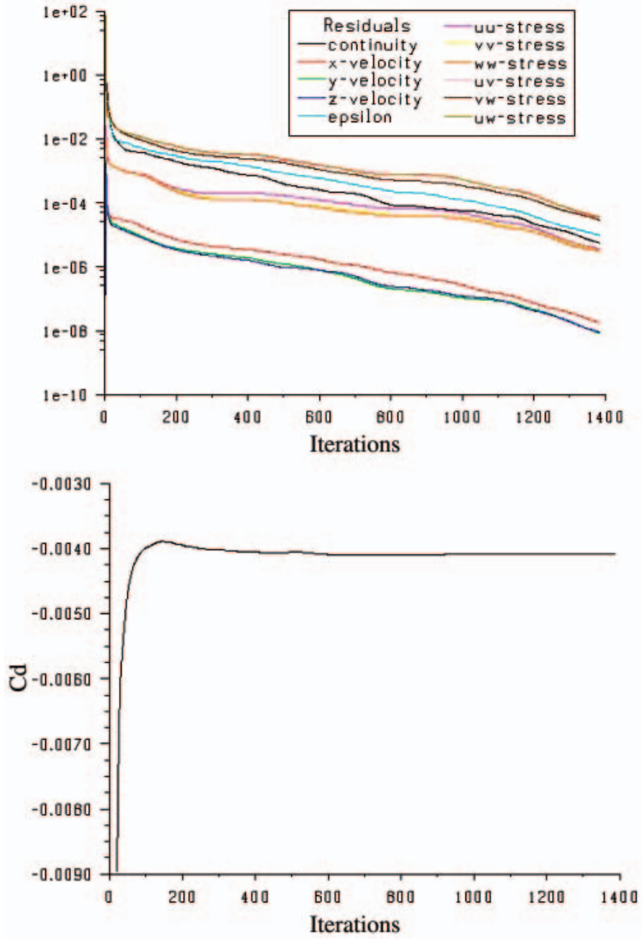

Fig. 4. Diagrams of iteration process convergence of resistance calculations of B573 ship's hull model

Comparison test for resistance calculations

Ship draught T = 11.3 [m]
Water density $\rho = 999.0$ [kg · m^{-3}]
Kinematic viscosity $\nu = 1.13896$ [$m^2 \cdot s^{-1}$]

The calculations were conducted for three speed values – their results are given in Tab. 2 and Fig. 5.

Tab. 2. Comparison test for resistance

Vs [knots]	Vm [m/s]	Model resistance acc. basin model tests [N]	Model resistance acc. numerical calculations [N]
11.00	1.132	34.795	36.18
14.50	1.492	59.307	61.58
16.00	1.646	76.500	73.5

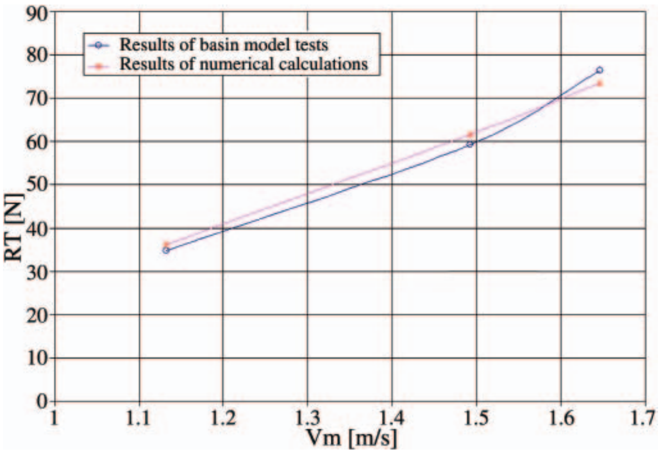

Fig. 5. Model resistance (comparison test)

NUMERICAL RESISTANCE CALCULATIONS FOR MODIFIED HULL VERSIONS OF B 573 SHIP

On the basis of the available hull documentation (hull body lines, Fig. 1) various hull form modifications were prepared. Their scope covered the following (Tab. 3):

- change of the ship's hull block coefficient CB
- change of the ship's hull prismatic coefficient CP
- change of location of the longitudinal centre of buoyancy LCB
- manual change of the entire ship's hull form or its stern part only.

Tab. 3. Range of changes of B573 ship hull form

Number of variant		Change of parameter	Parameter
1	Change of the block coefficient C_B; its initial value: $C_B = 0.800$	-0.01	0.790
2		-0.005	0.795
3		-0.015	0.785
4		+0.01	0.810
5		+0.005	0.805
6		+-0.015	0.815
7	Change of the prismatic coefficient C_P; its initial value: $C_P = 0.790$	-0.01	0.780
8		-0.02	0.770
9		-0.03	0.760
10		+0.01	0.800
11		+0.02	0.810
12	Change of longitudinal location of the centre of buoyancy LCB. its initial location: equal to 48% Lwl measured aft from FP	-1% shift fore	47%
13		-2% shift fore	46%
14		-3% shift fore	45%
15		+1% shift aft	49%
16		+2% shift aft	50%
17		+3% shift aft	51%
17	Manual hull form modification of its stern part only	-	-

The hull form modifications were so performed as to maintain the ship's displacement ∇ constant:

$$\nabla = L \cdot B \cdot T \cdot C_B \qquad (1)$$

and the ship's breadth as well. In the case of the analyzed ship the breadth amounted to 32.2 m, hence its enlargement would not be justified from the point of view of the imposed design limitations. While changing the parameters given in particular groups the remaining quantities were left unchanged.

For each of the hull versions new body lines and next numerical computational grids were prepared, and finally calculations of resistance and wake distribution were performed. Example frame sections of modified hull form variants are shown in Fig. 6 ÷ 9.

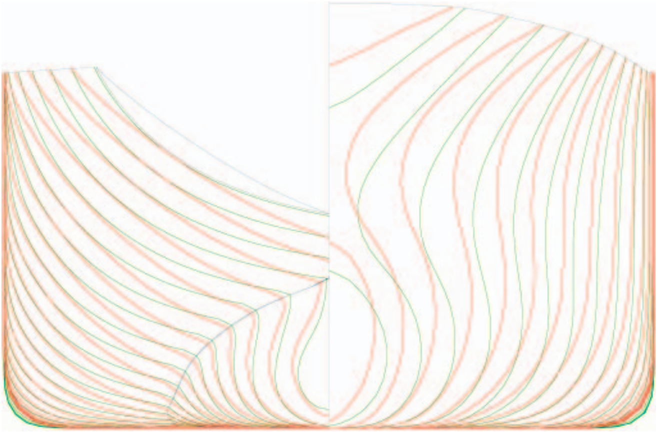

***Fig. 6.** Variant no.1 – original hull form - marked green; modified hull form – marked red.*

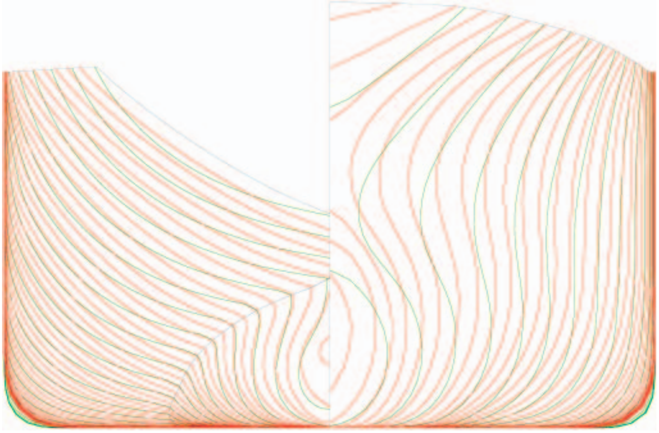

***Fig. 7.** Variant no.2 – original hull form - marked green; modified hull form – marked red.*

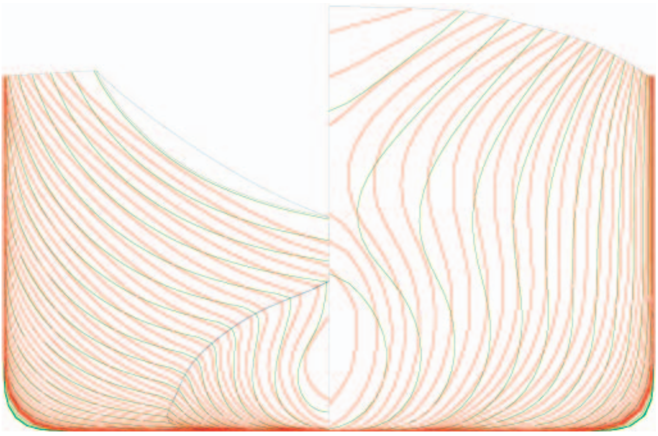

***Fig. 8.** Variant no.3 – original hull form - marked green; modified hull form – marked red.*

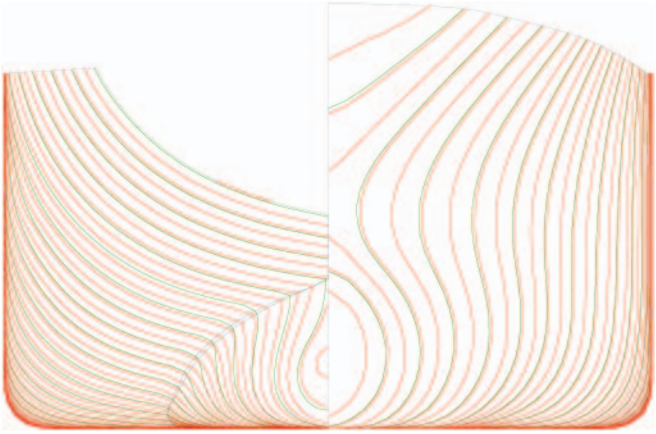

Fig. 9. *Variant no.18 – original hull form - marked green; manually modified hull form – marked red.*

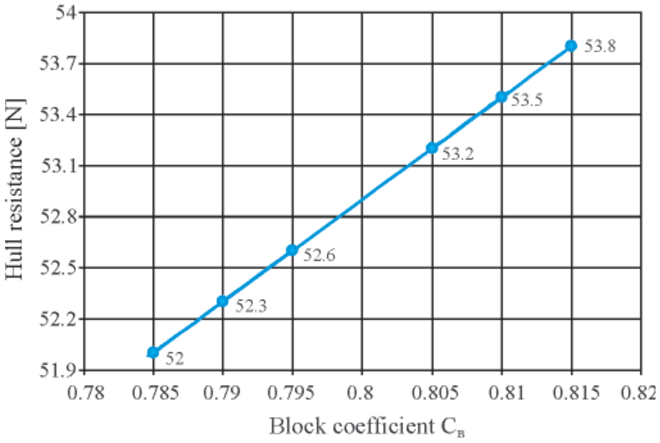

Fig. 10. *Calculated hull resistance in fuction of C_B (Vm=1.492 m/s)*

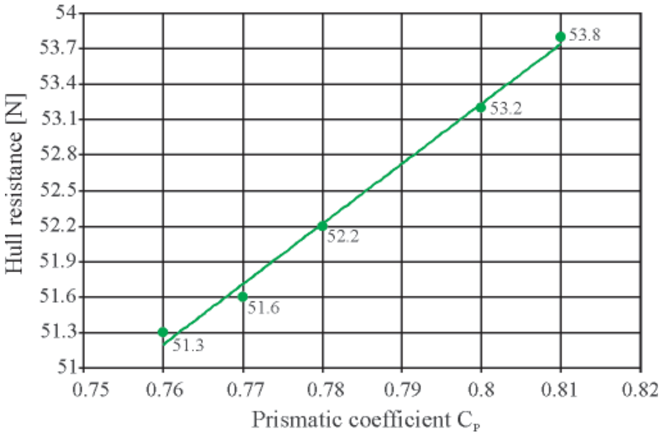

Fig. 11. *Calculated hull resistance in fuction of C_P (Vm=1.492 m/s)*

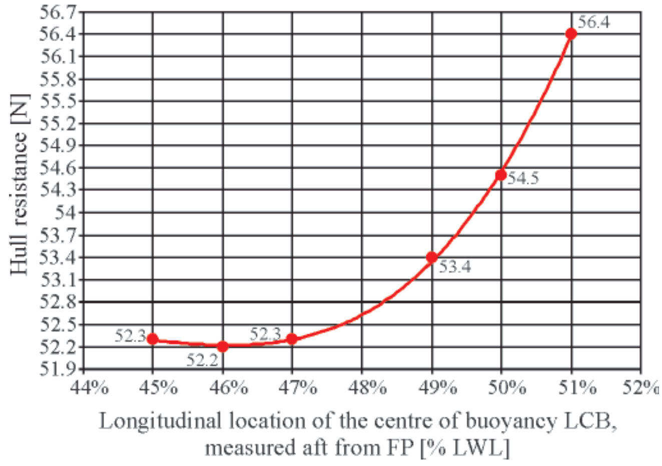

Fig. 12. *Calculated hull resistance in fuction of LCB (Vm=1.492 m/s)*

Tab. 4. *Results of numerical resistance calculations for modified form versions of ship hull model (V_m = 1.492 m/s)*

Number of variant	Modified parameter	Hull model resistance [N]
	C_B	
1	0.790	52.3
2	0.795	52.6
3	0.785	52.0
4	0.810	53.5
5	0.805	53.2
6	0.815	53.8
	C_P	
7	0.780	52.2
8	0.770	51.6
9	0.760	51.3
10	0.800	53.2
11	0.810	53.8
	LCB	
12	47%	52.3
13	46%	52.2
14	45%	52.3
15	49%	53.4
16	50%	54.5
17	51%	56.4
18	**Manual modification**	52.7

Next resistance calculations were performed for 18 versions of hull form modifications presented in Tab. 2, for the speed V_m = 1.492 m/s; their final results are given in Tab. 4. Also, influence of particular modified form parameters of hull model on its resistance to motion is presented in Fig. 10 ÷ 12. Full range of the results obtained from the numerical calculations is contained in the research project's report [10].

CONCLUSIONS DRAWN FROM THE PERFORMED INVESTIGATIONS

- The results of the comparison test (Tab. 2, Fig. 5) reveal certain differences between basin model tests and numerical calculations. They may result form the following reasons:
  - Hull model as well as numerical computational grid has been elaborated on the basis of two-dimensional drawing of body lines. The three-dimesional models based on it may hence differ to each other, in such tests three-dimensional ship hull models should be rather used.
  - The numerical resistance calculations have been performed for hull form model of B 573 ship without propeller, whereas basin model tests have been conducted with the use of a fairwater body substituting propeller cone.
  - Accuracy of numerical calculations may be influenced by a type of computational grid, number of computational domain elements as well as other parameters of CFD method, that should be further investigated.
  - The ship resistance numerical calculations were performed for the selected ship hence it is necessary to examine whether for other hull forms similar relations would be obtained.

- The results obtained from numerical resistance calculations for modified ship hull form versions are not any new relations and such trends are rather obvious. However it is interesting that the numerical method properly reproduces the trends, all the more that the changes of the parameters are relatively small (similar relationships have been also obtained from the approximate method [2] but only for varying global geometrical parameters since the method has not covered geometrical modifications of stern part of ship hull). This may serve as a basis for application of such approach to optimization of ship hull form and improvement of its propulsion efficiency. Especially, the diagram showing the relation between resistance and longitudinal location of buoyancy centre, which suggests existence of a minimum resistance for LCB relative value equal to 46%.

Acknowledgements

Results of the investigations presented in this paper have been financed by Ministry of Science and Higher Education in the frame of the research project no. R10 008 01.

NOMENCLATURE

B – hull breadth
c_B – hull block coefficient
c_P – hull prismatic coefficient
L_{CB} – longitudinal location of ship centre of buoyancy
L_{PP} – ship length between perpendiculars
L_{WL} – ship waterline length
R_T – ship hull resistance
S – area of hull wetted surface
T – ship draught
Vm – ship model speed
Vs – ship speed
∇ – volumetric displacement of underwater part of hull
ρ – water density
ν – water kinematic viscosity.

BIBLIOGRAPHY

1. Abt C., J.J. Harries S., Heimann J., Winter H.: *From Redesign to Optimal Hull Lines by Means of Parametric Modelling*, COMPIT 2003, Hamburg, May 2003
2. Harries S., Valdenzzi F., Abt C., Viviani U..: *Investigation on Optimisation Stretegies for the Hydrodynamic Design of Fast Ferries*, FAST01, Southampton, September 2001
3. Hoekstra M., Raven H.: *A practical approach to constrained hydrodynamic optimization of ships*, NAV 2003, Palermo, June 2003
4. Hollenbach K.U.: *Estimating resistance and propulsion for single-screw and twin-screw ship.* Ship Technology Research 45/2, 1998
5. Holtrop J., Mannen G.G.J: *An Approximate Power Prediction Method.* ISP, Vol. 29, No. 335, 1982
6. Janson C.E., Larsson L.: *A method for the optimization of ship hulls from a resistance point of view*, 21st Symposium on Naval Hydrodynamics, Trondheim, 1996
7. Jaworski S., Syrocki W.: Ship B 573: *Results of Model Tests-Resistance, Wake Measurements*. Technical Report No RH-95/T-041A, Ship Design and Research Centre, Gdańsk, 1995
8. Maisonneuve J.J., Harries S., Marzi J., Raven H.C., Viviani U., Piippo H.: *Towards Optimal Design of Ship Hull Shapes*, IMDC'03, Athens, May 2003
9. Marzi J.: *Use of CFD methods for hullform optimization in a model basin*, MARNET – CFD Workshop, Haslar, U.K., 2003
10. Szelangiewicz T.: *Numerical research on rudder-propeller -ship stern interaction, aimed at improving propulsion and manoeuvrability qualities of transport ship* (in Polish). Appendix to the final report on realization of the research project no. R 10 008 01, Szczecin, 2009.

CONTACT WITH THE AUTHORS
Tadeusz Szelangiewicz, Prof.
Tomasz Abramowski, Ph. D.
West Pomeranian University of Technology, Szczecin
Faculty of Marine Technology
Al. Piastów 41
71-065 Szczecin, POLAND
e-mail: tadeusz.szelangiewicz@ps.pl