UDC 629.5.015.4:629.543

Boris BUĆAN
Marta Pedišić BUČA
Stanislav RUŽIĆ

Numerical Modelling of the Flow Around the Tanker Hull at Model Scale

Preliminary communication

Principles of RANSE-based (Reynolds Averaged Navier-Stokes Equations) numerical modelling of a flow around the tanker model hull are introduced with the application of the CD-adapco STAR-CCM+ (Computational Continuum Mechanics) software package in the field of ship hydrodynamics. The numerical model used in simulation is drawn up through base parameters and methods of geometrical and mathematical modelling. Results of two numerical simulations performed at design and ballast loading conditions are presented and a comparison with available experimental data is made. Some aspects of further improvements of the presented numerical model are pointed out.

Keywords: *tanker ships, flow around the ship hull, numerical simulation, RANSE, model tests, experiment*

Authors' address:
Numerical Hydrodynamics Department, Ship Hydrodynamics, Brodarski institut d.o.o., Av. V. Holjevca 20, 10 000 Zagreb, Croatia
E-mail: boris.bucan@hrbi.hr; marta.pedisic@hrbi.hr; stanislav.ruzic@hrbi.hr
Phone: +385 (0)1 6504 273
Fax: +385 (0)1 6504 230

Received (Primljeno): 2008-04-18
Accepted (Prihvaćeno): 2008-05-12
Open for discussion (Otvoreno za raspravu): 2009-06-30

Numeričko modeliranje strujanja oko trupa tankera u modelskom mjerilu

Prethodno priopćenje

Predstavljena su načela RANSE (Reynolds Averaged Navier-Stokes Equations) numeričkog modeliranja strujanja oko modela trupa tankera, primjenom programskog paketa STAR-CCM+ (Computational Continuum Mechanics) tvrtke CD-adapco u području brodske hidrodinamike. Opisani su osnovni parametri i metode geometrijskog i matematičkog modeliranja korištenog numeričkog modela. Prikazani su rezultati dvije numeričke simulacije provedene za projektno i balastno stanje opterećenja modela tankera, te je komentirana usporedba s dostupnim podacima eksperimentalnog ispitivanja. Istaknute su neke od mogućnosti daljnjeg poboljšanja prikazanog numeričkog modela.

Ključne riječi: *tankeri, strujanje oko brodskog trupa, numerička simulacija, RANSE, modelska ispitivanja, eksperiment*

1 Introduction

Witnessing the rapid growth of the low-cost desktop computational power and access to high performance computing platforms on one side, and development in theoretical continuum mechanics methods and their numerical implementation on the other, one must recognize its strong impact in everyday engineering practice [1]. Following this trend, numerical modelling of the real complex physical phenomena is transforming from approximate, economically unaffordable, time-consuming applications limited mainly to critical parts of product development into a reliable and common design office tool, fully integrated into the overall product development process.

In that light, driven by the need for delivering on-time, on-cost cutting edge technology products and services, *Brodarski institut d.o.o.* (*BI*) has established a new Numerical hydrodynamics department with the main objective to increase the level of its competitiveness on the global market. At the first stage of department operability, the evaluation of few commercial numerical codes for general continuum mechanics is made from several relevant aspects of engineering usage. Recent work of the department regarding the quantitative validation of the numerical results performed in STAR-CCM+ with reference to experiment is presented in this paper.

2 Benchmark

A hull form of a tanker model extensively tested in BI's towing tank is chosen, providing a set of reference experimental data for a comparison [2]. A quantitative margin of ±10% to measured data is adopted as reasonable in the first iteration of computations, leaving a considerable space for further improvements of the numerical model by means of higher accuracy and reduction in computation time.

Two simulations are performed on the bare tanker model at design and ballast loading conditions (Table 1), each at a single model speed of V=1.419 m/s corresponding to Froude numbers based on L_{WL} Fr=0.176/0.178 and Reynolds numbers of Re=8.67x10^6/8.38x10^6 respectively.

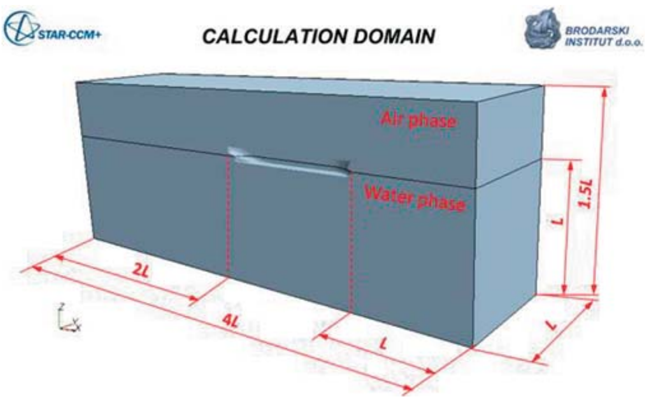
Table 1 **Tanker model particulars**
Tablica 1 **Karakteristike modela tankera**

		Design	Ballast
Length of waterline	L_{WL}	6.6608 m	6.4538 m
Breadth, moulded	B	0.9578 m	0.9578 m
Draught at FP	T_F	0.3624 m	0.2228 m
Draught at AP	T_A	0.3624 m	0.2377 m
Displacement volume	∇	1.9478 m^3	1.1880 m^3
Wetted surface area	S_0	9.8870 m^2	7.8634 m^2

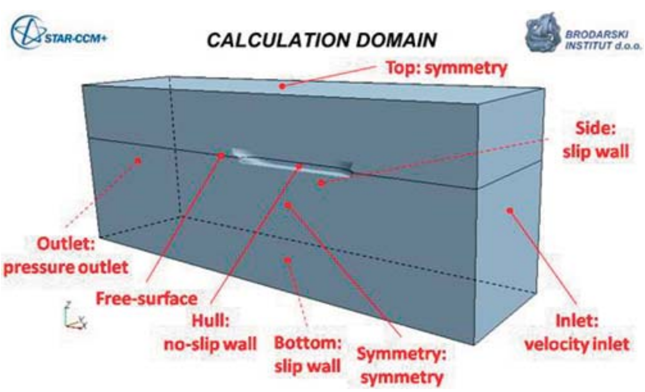
3 Geometrical model setup

STAR-CCM+ uses a finite volume method based solver [3]. The governing equations describing the physical model of the flow are discretized in the number of control volumes (CV), or cells, in which the unknown variables are solved. A 3-D rectangular computation domain representing a part of the towing tank is used in both simulations. Due to the vertical plane symmetry, only a half of the computation domain is modelled. The domain length is taken up to four model lengths, one length in front and two lengths behind the hull. Furthermore, the domain is modelled one model length wide and one and a half model length high, having a free-surface at one model length from the bottom, defining the water depth (Figure 1a).

Figure 1 **Calculation domain; a – extents, b – boundary conditions**
Slika 1 **Proračunska domena; a – dimenzije, b – rubni uvjeti**



a)



b)

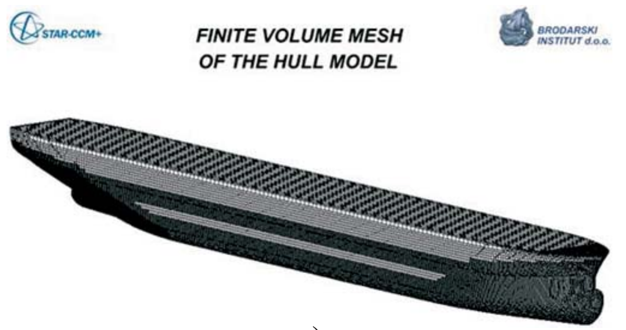
The process of the finite volume mesh generation starts with a definition of the core mesh surfaces and their triangulation. The core mesh is a rectangular portion of the computational domain that closely surrounds the tanker model and which is modelled with most care. At this point, the user is allowed to define the variable triangle sizes on all of the surfaces defining the core mesh. Afterwards, the trimmed cell method is applied to generate volume cells inside the core mesh according to user settings. Finally, the core mesh boundaries are extruded to the extents of the computation domain.

Finite volume meshes for both loading conditions are generated entirely with hexahedral cells. Parameters used in the generation of the meshes are identical for both loading conditions (Table 2), with the exception of the total number of generated cells due to the difference in tanker model position relative to the free-surface (Figure 2). Cell sizes in the domain are gradually refined towards the regions where high gradients of velocity and pressure are expected. A prismatic layer is generated around the hull model, defining the appropriate cell thicknesses in order to resolve the boundary layer.

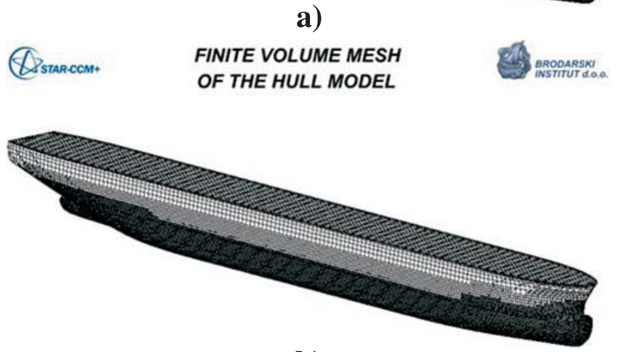
Table 2 **Parameters of finite volume meshes**
Tablica 2 **Parametri mreže konačnih volumena**

	Design	*Ballast*
Total number of generated cells	2447012	1904370
Max. cell size on hull (water phase)	0.0125 m	0.0125 m
Max. cell size on hull (air phase)	0.05 m	0.05 m
Max. cell size on domain boundaries	0.2 m	0.2 m
Max. cell size on free-surface	0.0125 m	0.0125 m
First cell thickness (water phase)	0.0025 m	0.0025 m
First cell thickness (air phase)	0.005 m	0.005 m

Figure 2 **Finite volume mesh of the tanker model; a – design draught, b – ballast draught**
Slika 2 **Mreža konačnih volumena modela tankera; a – projektni gaz, b – balastni gaz**



a)



b)

4 Physical model setup

RANSE solver is used to solve the governing equations of continuity and momentum in a viscous flow around the hull model. The Navier-Stokes Equations (NSE) are solved with a segregated, algebraic multigrid solver [4] using the Rhie-Chow interpolation for pressure-velocity coupling [5]. The SIMPLE algorithm [6] is a guess-and-correct procedure applied to control the overall solution. Starting from the boundary conditions the velocity and pressure gradients are calculated and the discretised momentum equations are solved. That gives the intermediate velocity field which is used to calculate the uncorrected mass fluxes at the cell faces. After correcting the pressure field (p) and the boundary pressures, the mass fluxes across the cell faces can be corrected and the cell velocities (u, v, w) can be updated.

To obtain the RANSE for a turbulent flow around the ship hull, the NSE for the instantaneous velocity and pressure fields are decomposed into a mean and a fluctuating component and the averaging process may be thought of as time averaging for steady-state situations and ensemble averaging for repeatable transient situations. The effect of turbulent fluctuations on the mean flow is approximated with a set of semi-empirical expressions called turbulence model. The Menter shear-stress transport (SST) k-ω turbulence model [7, 8] is used in the simulations, in which the transport equations are solved for two variables: turbulent kinetic energy (k) and specific dissipation rate (ω).

Free-surface is modelled using the Volume of Fluid approach (VOF) with a High Resolution Interface Capturing Scheme (HRIC) based on the Compressive Interface Capturing Scheme for Arbitrary Meshes (CISCAM) [9, 10]. The numerical model can be applied to any structured and unstructured grid with arbitrary shaped CV. With four conservation and two turbulence equations an additional equation is solved, introducing a variable c ($0 \leq c \leq 1$) for the volume fraction of the each fluid, where 1 stands for filled CV of one fluid in which at the same time the volume fraction of other fluid has to be 0 to achieve unity. This approach treats both fluids as a single fluid which changes its properties according to the volume fraction.

The boundary condition applied to the model hull is a no-slip wall allowing for boundary layer around the model hull to develop. All-y^+ wall treatment is used for a set of near wall modelling assumptions. Bottom and side boundaries are modelled as slip walls, while top and symmetry boundaries are modelled as symmetry. The velocity inlet and the pressure outlet are applied to inlet and outlet boundaries (Figure 1b).

5 Numerical results and comparison with experiment

The transient simulations are stopped when the difference in successive residuals of the pressure drag coefficient over a specific time period at any point on the hull model reaches the maximum value of 10^{-4}, where the specific time period refers to the time needed for a particle of water to travel one tanker model length (approximately 5 s). Mean values of the drag, the shear and the pressure component of the drag, and the lift forces over a specific time period, are calculated for comparison with experimental data (Table 3).

Table 3 **Comparison of the force coefficients**
Tablica 3 **Usporedba koeficijenata sila**

Design	Computed	$10^3 x C_D$	$10^3 x C_{DS}$	$10^3 x C_{DP}$	C_L
		3.897	2.954	0.944	1.862
	Measured	$10^3 x C_T$	$10^3 x C_{F0}$	$10^3 x C_R$	C_W
		3.947	3.076	0.871	1.920
	Margin	-1.3 %	-3.1 %*	+1.8 %*	-2.3 %
Ballast	Computed	$10^3 x C_D$	$10^3 x C_{DS}$	$10^3 x C_{DP}$	C_L
		4.409	3.094	1.314	1.428
	Measured	$10^3 x C_T$	$10^3 x C_{F0}$	$10^3 x C_R$	C_W
		4.316	3.094	1.222	1.472
	Margin	+2.1 %	0.0 %*	+2.1 %*	-3.0 %

* Absolute margins are presented in correlation with total force coefficients.

Where

C_D... drag coefficient
C_{DS} ... shear drag coefficient
C_{DP} ... pressure drag coefficient
C_L ... lift coefficient
C_T ... total resistance coefficient
C_{F0} ... frictional resistance coefficient of a corresponding plate (ITTC-1957 correlation line)
C_R ... residual resistance coefficient
C_W ... weight coefficient.

Results of the free-surface elevation computation around the tanker model at the design and the ballast loading condition are presented qualitatively, and compared with photographs taken during the testing of the model in the towing tank (Figures 3 and 4). Development of the bow and stern wave systems are captured realistically at both loading conditions, forming a wave patterns around the hull model similar to the ones observed in the experiment.

Streamlines plotted on the hull model indicate a possible flow separation at the stern bulb region near the propeller, at both loading conditions (Figures 5b and 5d). A further analysis of the shear stress on the hull model reveals the presence of reverse flow region at the same position (Figure 6a). This result is observed in the experiment performed at design draught as well (Figure 6b).

Wake fields computed at propeller disk for both loading conditions are compared to the experiment by overlapping the measured black-and-white iso-wake curves over computed colour-filled results (Figure 7). Qualitatively, shapes of the wake fields are very well captured at both loading conditions. On the other hand, computed axial components of velocity over propeller disk locally show the absolute margin of up to 0.15 to measurements.

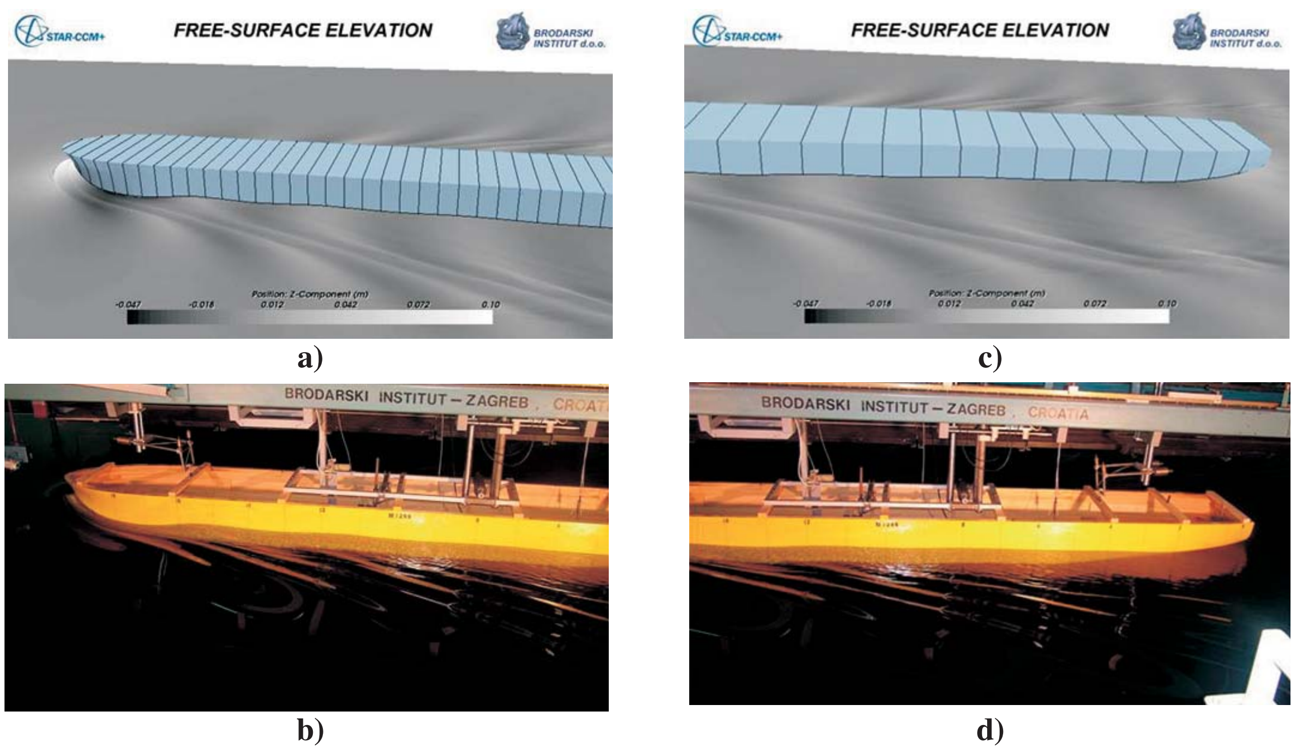
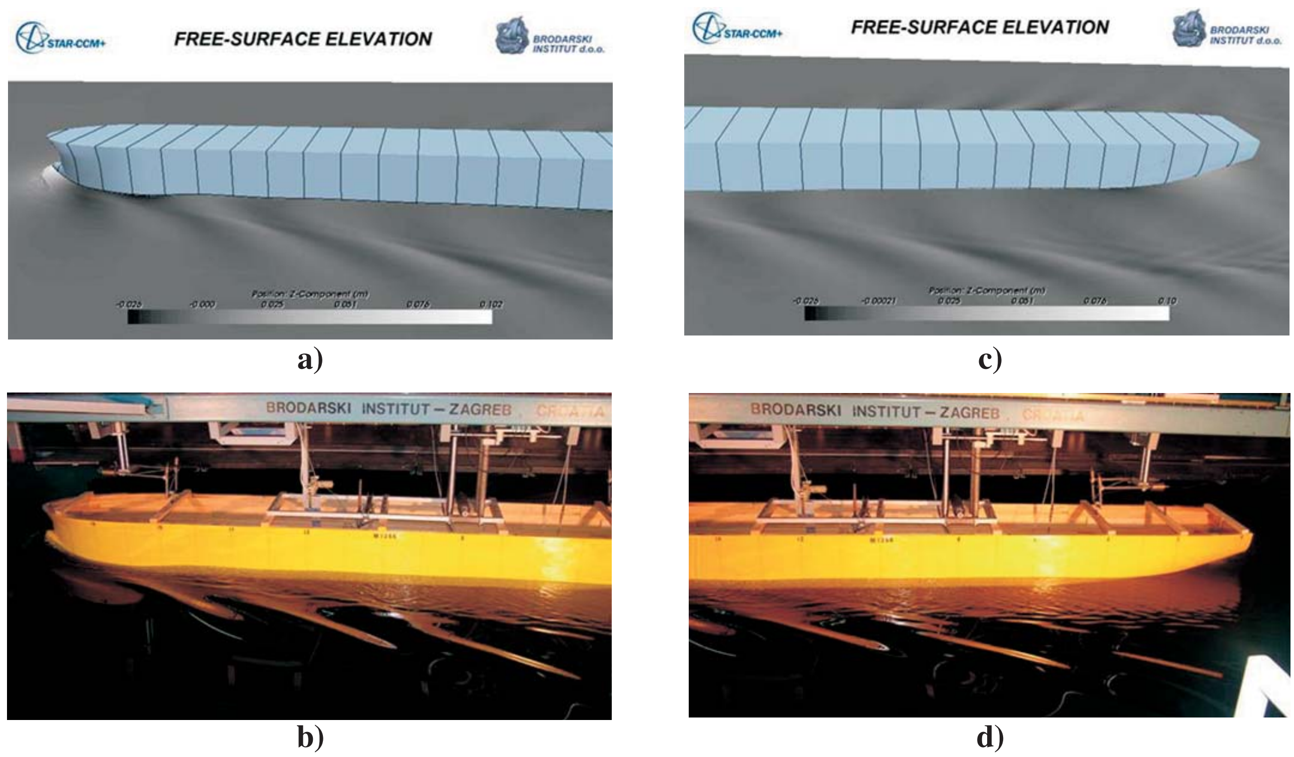


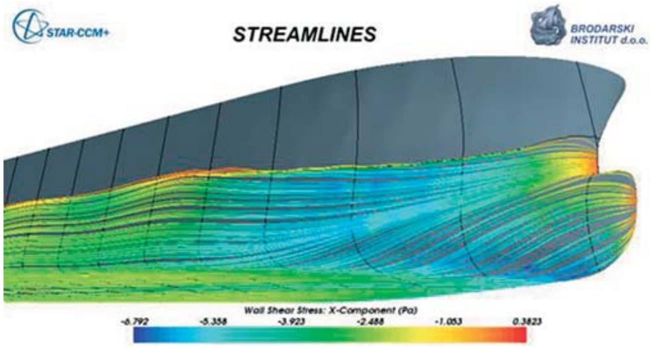
Figure 3 **Comparison of the free-surface elevation around the tanker model at design draught; a, c – numerical computation; b, d – photograph taken during the experiment**

Slika 3 **Usporedba elevacije slobodne površine oko trupa modela tankera na projektnom gazu a, c – numerički proračun; b, d – fotografija snimljena tijekom eksperimenta**

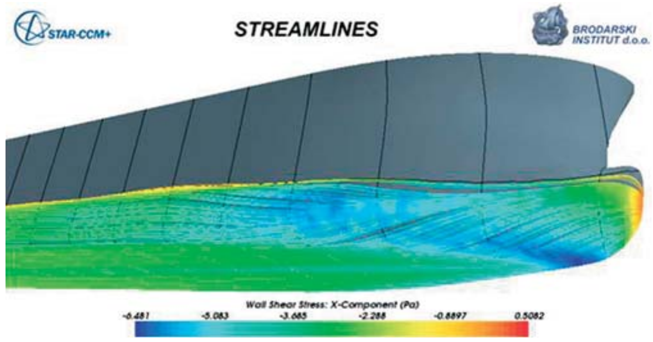
Figure 4 **Comparison of the free-surface elevation around the tanker model at ballast draught a, c – numerical computation; b, d – photograph taken during the experiment**

Slika 4 **Usporedba elevacije slobodne površine oko trupa modela tankera na balastnom gazu a, c – numerički proračun; b, d – fotografija snimljena tijekom eksperimenta**

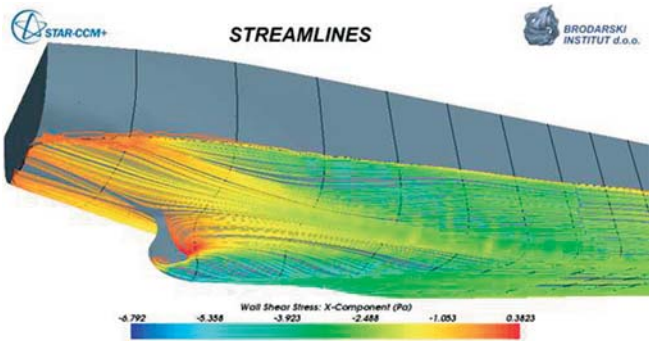




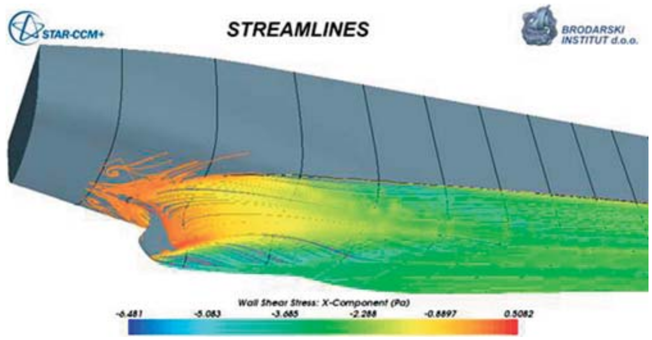
a)



c)



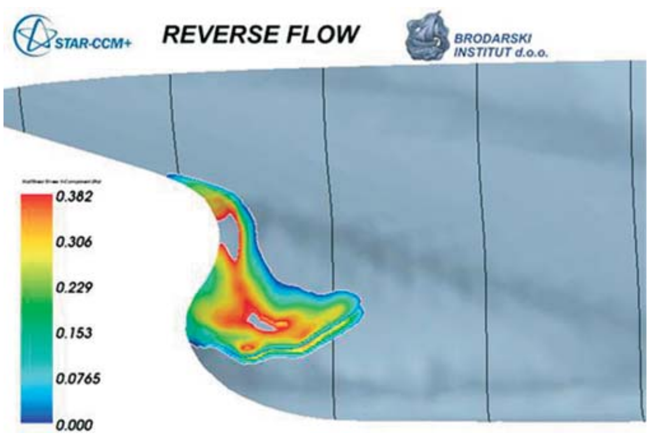
b)



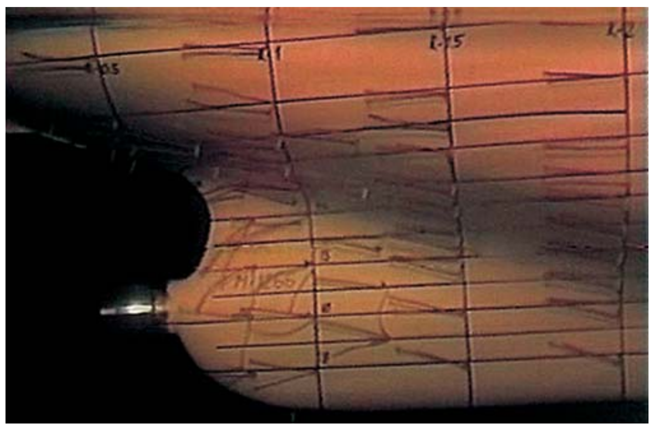
d)

Figure 5 **Streamlines on the tanker model hull; a, b – design draught; c, d – ballast draught**
Slika 5 **Strujnice izračunate na trupu modela tankera; a, b – projektni gaz; c, d – balastni gaz**

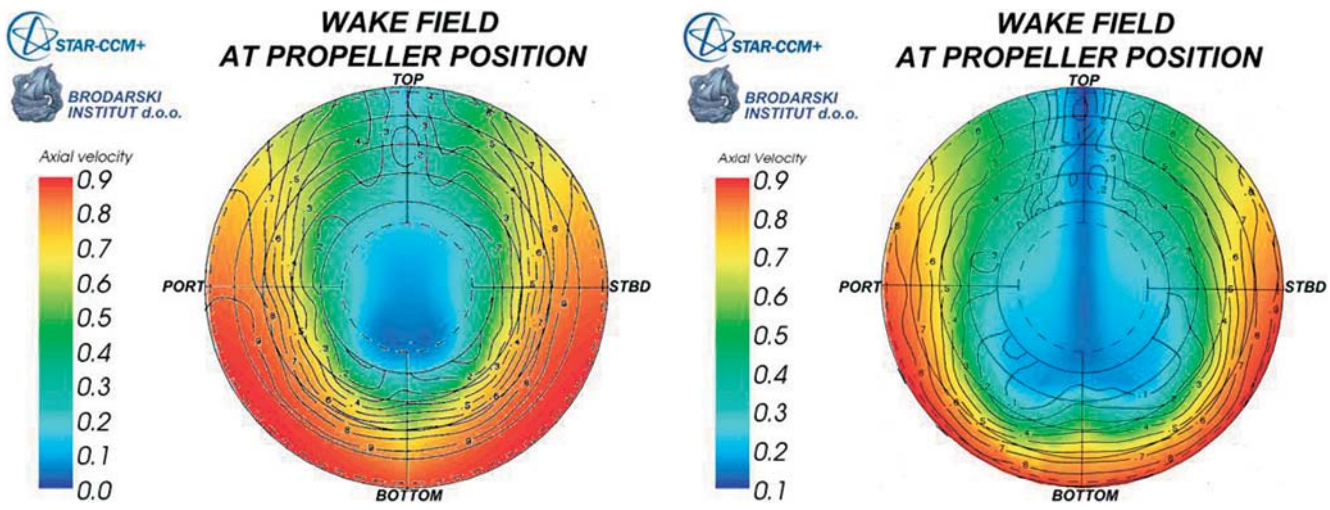
Figure 6 **Flow separation region at the stern of the tanker model;**
a – numerical computation; b – photograph taken during the experiment
Slika 6 **Područje odvajanje strujanja na krmi modela tankera;**
a – numerički proračun; b – fotografija snimljena tijekom eksperimenta



a)



b)



a) **b)**

Figure 7 **Comparison of the wake field at propeller position; a – design draught; b – ballast draught**
Slika 7 **Usporedba polja sustrujanja na poziciji vijka; a – projektni gaz; b – balastni gaz**

5 Conclusions

As to the results of the numerical modelling of the flow around the tanker model and the comparison with experimental data presented in this paper, the following can be concluded:

1) Values of computed drag and lift coefficients are satisfactory for both loading conditions, with a margin of ±3% to the experiment. It must be noted, however, that the comparison of coefficients of the horizontal force components presented in the Table 3 is subjected to the method used in the decomposition of the total resistance measured in the experiment.
2) The elevations of the free-surface calculated around the tanker model at both loading conditions are captured realistically, according to photographs taken during the experiment.
3) Flow separation is detected in the stern region of the tanker model near the propeller, at the same position as observed in the experiment.
4) Wake fields at propeller disk are calculated for both loading conditions within absolute margin of 0.15 to the experiment. Nevertheless, due to strong time-dependent turbulent fluctuations behind the tanker model hull, the computed results are acceptable, providing a valuable insight into the flow behind the model hull.

Although the obtained computational results in the scope of the available experimental data are satisfactory from quantitative aspect, investigation on further improvements of the presented numerical model can be performed. The geometrical model can be optimized regarding the total cell number, cell sizes and cell distribution in the calculation domain. An analysis of turbulence modelling can be performed with the use of more accurate turbulence models and an analysis of different wall treatments can be carried out.

References

[1] DEJHALLA, R., PRPIĆ-ORŠIĆ, J.: “A review of the state-of-the-art in marine hydrodynamics”, Brodogradnja, 57(2006)1, Zagreb, pp. 13-22

[2] SEMIJALAC, G.: “Resistance, self-propulsion, 3D wake and streamlines test results”, Brodarski institut d.o.o., Report No.: 5776-M, Zagreb, 2004.

[3] CD-adapco Group: “STAR-CCM+ Documentation”, CD-adapco Group, London, 2007.

[4] FERZIGER, J.H., PERIĆ, M.: “Computational methods for fluid dynamics”, 3rd rev. ed., Springer-Verlag, Berlin, 2002.

[5] RHIE, C.M., CHOW, W.L.: “A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation”, AIAA Journal, Vol. 21, 1983., pp. 1525-1532

[6] PATANKAR, S.V.: “Numerical heat transfer and fluid flow”, McGraw-Hill, New York, 1980.

[7] WILCOX, D.C.: “Turbulence modelling for CFD”, 2nd edition, DCW Industries Inc., 1998.

[8] MENTER, F.R.: “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA Journal, Vol. 32(8), 1994., pp. 1598-1605

[9] UBBINK, O.: “Numerical prediction of two fluid systems with sharp interfaces”, Ph. D. Thesis, University of London, 1997.

[10] MUZAFERIJA, S., PERIĆ, M.: “Computation of free surface flows using interface-tracking and interface-capturing methods”, Chap.2 in O. Mahrenholtz and M. Markiewicz (eds.), Nonlinear Water Wave Interaction, Computational Mechanics Publication, WIT Press, Southampton, 1999.