

Numerical Simulation of the Self-Propulsion Model Tests

Tomasz Bugalski, Pawel Hoffmann¹

¹Ship Design and Research Centre S.A. (CTO), Gdansk, Poland

ABSTRACT

One of the current research projects at the Ship Design and Research Centre S.A. (CTO) focuses on the investigation of flow around ship hull with rotating propeller. The goal of the project is to test and demonstrate the capabilities of Reynolds-averaged Navier–Stokes equations (RANSE) solver in the area of complex ship flow simulations. Focus is on a complete numerical model for hull, propeller and rudder that can account for the mutual interaction between these components when the flow field is computed. In the present work the real propeller geometry is modeled. The computations required using the “sliding grid” method – a mesh divided into two regions connected with an interface surface. The first region was a cylinder built around the propeller. The second one contained the rest of the computational domain. The mesh around the propeller rotates relative to a fixed mesh connected with the hull.

This paper presents the results of a complex investigation of the flow computations around the research/training vessel Navigator XXI and KRISO Container Ship (KCS). Meshing and flow simulations were conducted with StarCCM+ from CD-adapco.

Keywords

Ship propulsion, hull –propeller and -rudder interaction, wake field, CFD

1 INTRODUCTION

Hydrodynamic aspects play a significant role in the quality of a ship. Dominant criteria in the hull form design are resistance and powering performance, as well as the occurrence of noise and vibrations, which are important for the comfort level of crew and passengers. They often have a hydrodynamic cause, stemming from the operation of the propeller in the flow field behind the ship hull. Therefore, acquiring detailed knowledge of the flow around the aft part of a ship is critical. In this area, flow separation may occur, which would have important consequences for the resistance and power, and thus, the flow field felt by the propeller should be determined. Unfavorable characteristics of this flow field (e.g., large spatial variations of the velocity) may result in performance loss, propeller cavitation leading to noise, vibrations and erosion, etc. The viscous flow computed

using RANSE solvers has become a common component in the ship design process, and satisfactory agreement with model-scale wake field measurements can now be achieved. The real purpose is to predict the hull/propeller/rudder interaction, wake field and finally full-scale flow about ship hull. RANSE computations offer that possibility, and such viscous-flow computations have become progressively used in practical ship design; but how accurate these predictions are, and the extent to which they depend on the turbulence modeling being used, is not really known. A validation of the full-scale ship viscous flow predictions has been generally insufficient so far, mainly due to the virtual absence or difficult availability of the suitable full-scale experimental wake field data. CFD methods have been used in Poland for thirty years for studies on the nature of propeller-hull interaction. The methods developed during this time can be found in Bugalski & Jaworski (1984), Bugalski (1997) and Bugalski & Szantyr (1998). These studies have changed their character in light of the dynamic development of the CFD methods, but the original approach to the interaction phenomena still remains essential. The results of preliminary numerical simulation of the flow around KCS ship and KP505 rotating propeller were presented at Gothenburg 2010 – A Workshop on Numerical Ship Hydrodynamics. The computations were carried out in model scale in order to enable direct comparison of the CFD results with the experimental results (Bugalski & Hoffmann 2010).

2 MODEL GEOMETRY AND SIMULATION CONDITIONS

The non-standard model tests for the training/research vessel “Navigator XXI” have been performed at the Ship Hydromechanics Division of CTO in the frame of the European Full-Scale Flow Research and Technology EFFORT project (Bugalski & Kraskowski 2006). The training/research vessel “Navigator XXI” is operated by the Maritime University of Szczecin. The study was conducted on both the full scale ship and the model scale.

The KCS was conceived to provide data for both the explication of flow physics and the CFD validation for a modern container ship with a bulbous bow. The Korea Research Institute for Ships and Ocean Engineering (now MOERI) performed towing tank experiments to obtain

resistance, mean flow data and free surface waves (Van et al 1998, Kim et al 2001). Self-propulsion tests were carried out at the Ship Research Institute (now NMRI) in Tokyo and have been reported in the Proceedings of the CFD Workshop Tokyo in 2005 (Hino 2005). Data for pitch, heave, and added resistance are also already available from Force/DMI measurements reported in Simonsen et al (2008). No full scale ship exists. Main characteristics of the hull shapes, scale factors and visualizations of the hull shapes are presented below in Table 1.

Table 1: Main data of the ships

	Training ship	Container ship
Ship	NawigatorXXI	KCS
Length b.p. [m]	54,13	230,00
Length of waterline [m]	55,16	232,50
Breadth [m]	10,50	132,20
Draught [m]	3,15/3,20	210,80
Displacement [m ³]	1126	52030
Surface wetted area [m ²]	672	9424
Block coefficient	0,623	0,651
Midship section coefficient	0,915	0,985
Service speed [kn]	9,98	24,0
Propeller	CP469	KP505
Rotation	Left	right
Diameter [m]	2,26	7,90
Number of blades	4	5
Expanded area ratio	0,6728	0,7963
Pitch ratio	0,9424	1,1740
Blade rake	0	0
Skew back [deg]	28	0
Profile	NACA16/0.8	NACA66/0.8

As mentioned above, the computations were carried out at model scale. The scale factors are the same as for the models used for towing tank experiments, and are listed in Table 2. The scaled values of speed are also given.

Table 2 Scale factors and speeds in model scale

Ship	Training ship	Container ship
Scale factor	10,00	31.60
Model speed [m/s]	31.609	42.196

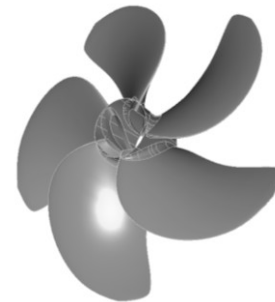
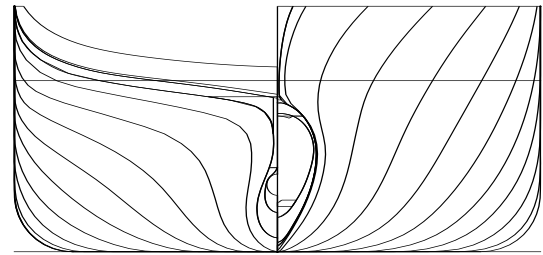


Figure 1: Geometry of the KCS hull and KP505 propeller

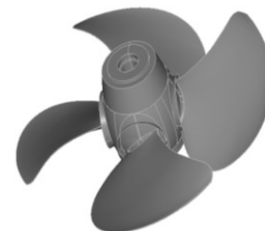
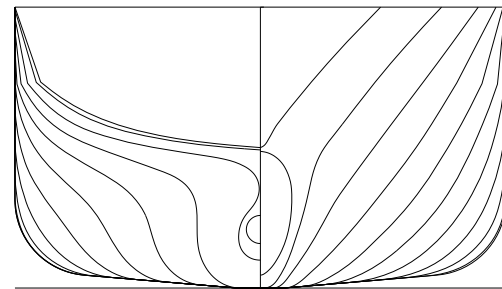


Figure 2: Geometry of the Nawigator XXI hull and CP469 propeller

3 NUMERICAL METHOD FOR SIMULATION OF SHIP SELF-PROPULSION TESTS

The computations are performed with the Reynolds Averaged Navier-Stokes (RANSE) solver StarCCM+ from CD-adapco. The code solves the RANSE and continuity equations in integral form on a polyhedral mesh by means of the finite volume technique. Both steady state and transient calculations are considered. The Reynolds stress problem is solved by means of k-ε turbulence

model. The rotating propeller is treated in the following different ways:

- For open-water calculations the propeller inflow is uniform, so the moving reference approach is applied;
- For the propeller rotating behind a ship, a rigid body approach is applied.

The free surface is modeled with the two phase volume of fluid technique (VOF).

The RANSE method is based on the fundamental conservation equations. However they are solved for the values averaged in a time range longer than the period of velocity and pressure fluctuations due to turbulence. Direct numerical flow modeling requires excessive computational effort due to the requirement of very dense discretization of computational domain and use of very short time step. It is assumed that the influence of turbulence on the flow is derived with sufficient accuracy based on the averaged flow parameters. Therefore, the additional, partly empirical relationships between factors characterizing the turbulence – the so-called Reynolds stress and averaged flow parameters – have been introduced. They are referred to as so-called turbulence models. Because turbulence modeling is by design a kind of simplification and because universal relations for proper turbulence modeling of all flow types have not yet been formulated, different turbulence models are applied, depending on a requested accuracy and a flow type. In case of the present computations, “k-epsilon” model is applied: It is a standard model of high stability and accuracy level, sufficient for most of engineering works. There are many variants of this model; “realizable k-epsilon” is used here.

Equations of the fluid motion are solved numerically using the finite volume method. This method can be described in brief as follows:

The flow domain is divided into small cells called finite volumes. The fluid motion equations are integrated numerically for each finite volume having the values of flow parameters attributed to its center. Equations of the fluid motion integrated for all the finite volumes compose a system of algebraic equations which is solved iteratively. The transformation of analytical equations into algebraic equations for a finite number of points in the space is called their discretization. The aim of the discretization is to set up the equations in such way that unknowns are values in central points of cells; therefore, it is required to express the values on the cells’ faces as a function of their central points.

There are many ways for calculation of values on faces based on the cell’s central point values. In the current instance, the so-called Second-Order Upwind scheme is used.

3.1 Open water propeller simulation

The open water calculation was carried out at the same running conditions as used in the experimental set up. The solution domain was chosen to extend 10 propeller

diameters in front of the propeller, 3 diameters in the radial direction and 5 diameters behind the propeller.

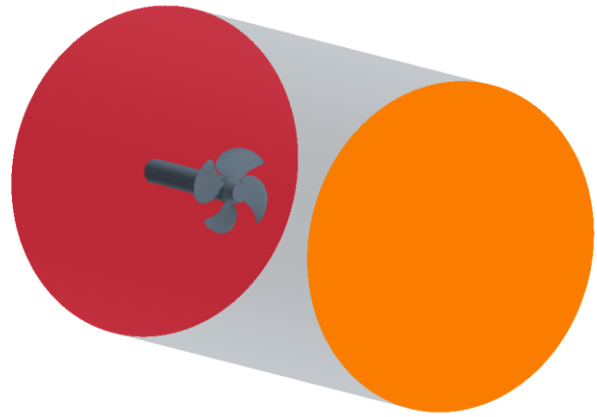


Figure 3: Computational domain in open water propeller test simulation

The flow solver was running in steady mode and the rotation of the propeller was accounted for by using the moving reference frame approach. This approach works fine in open water, where the propeller experiences a completely uniform inflow field. The calculations were carried out for advance coefficients in the range from $J=0.0$ to $J=1.0$. Similar to the experiment the computed thrust and torque on the propeller were converted into the dimensionless thrust coefficient, torque coefficient and the efficiency was calculated. The study of the flow field shows that the phenomena occurring in the flow (pressure distribution, tip vortices etc.) are typical for propeller flow and can be considered qualitatively correct. The comparison between the calculated and measured data for P505 –KCS propeller, shown in the Figure 4, proves that fairly good agreement is achieved. Particularly in the region around $J=0.6-0.8$, which corresponds to the usual propeller operation.

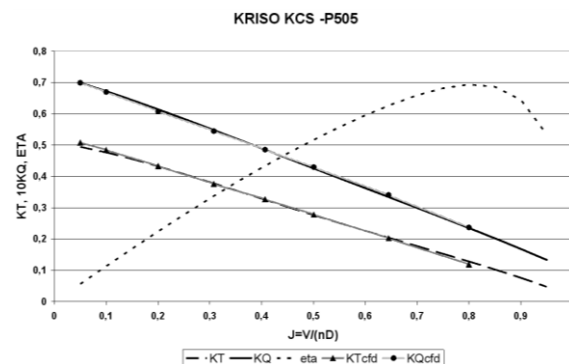


Figure 4: Comparison of open water hydrodynamic characteristics of KP505 propeller computed and obtained from experiment

3.2 First Self –propulsion Simulations (KCS)

The first goal of the project was to simulate numerically the flow about KCS ship hull and propeller including the

free surface around the ship. The calculations have been performed for the KCS ship without rudder in fixed conditions (suppressing dynamic sinkage and trim).

The numerical simulation of KCS self-propulsion test was developed in three steps: simulation of the open water propeller test, computations of flow around ship hull without propeller (equivalent to resistance test) and computation of the flow around ship with working propeller (equivalent to propulsion test). The basic assumptions for the simulation were as follows: The dynamic trim and sinkage of the hulls were neglected, propeller operation was modeled.

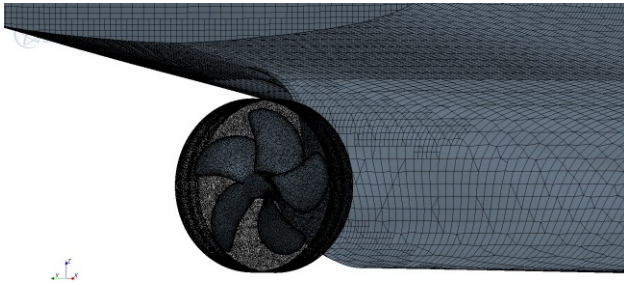


Figure 5: Mesh in the stern part of ship hull with propeller

The unstructured computational mesh of hexahedral and polyhedral cells was used for the computations. The results of study of grid size and turbulence model influence on ship hull without propeller resistance are presented in Table 3.

Table 3: Influence of grid size & turbulence models on KCS ship model resistance

Grid type	Turbulence model	Resistance [N]
Coarse (950k)	k- ω	41,0
	k- ϵ	41,8
	RSM	42,0
Medium (1,7M)	k- ϵ	41,8
Fine (2,6M)	k- ϵ	41,8

Finally, the total number of grid elements used in the calculation was 2,03 million. The example of mesh details are presented in Figure 5.

The self-propulsion computation has been carried out at the ship self-propulsion point following the experimental procedure. Thus, the rate of revolutions of the propeller n has been adjusted to obtain force equilibrium in the longitudinal direction considering the applied towing force (Skin Friction Correction, SFC), i.e.

$$T = RT(SP) - SFC$$

where T is the computed thrust, $RT(SP)$ is the total resistance at self-propulsion and $SFC = 30.3$ N (from the tests). The simulation with the propeller behind the ship must be run in transient mode, i.e. time accurate.

Comparison of experimental data (EFD) and computational results for the ship hull and rotating propeller are listed in Table 4.

Table 4: Verification and validation (V&V) study for FX0, KCS self-propulsion prediction, $Re = 1.4 \times 10^7$, $Fr = 0.26$

Parameters		EFD(D)	V&V Study
CT*103	Value	3,966	3,804
	E%D		4,08%
KT	Value	0,1700	0,1502
	E%D		11,65%
KQ	Value	0,0288	0,0283
	E%D		1,77%
n (rps) (for given SFC)	Value	9,50	9,80
	E%D		-3,16%
(RT(SP)-T) [N] (for given n)	Value	30,25	30,20
	E%D		0,17%

The axial velocity contours and cross flow vectors downstream of propeller plane ($x/LPP=0.9911$) are compared with the experimental data (Hino 2005), as shown in Fig. 6 and 7.

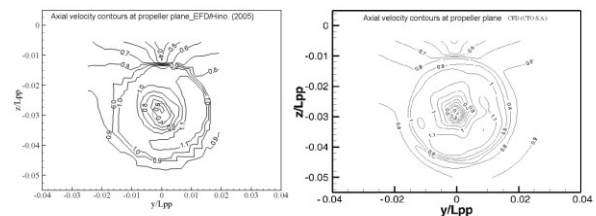


Figure 6: Comparison of axial velocity contours downstream of propeller plane ($x/LPP=0.9911$) EFD –left figure, CFD – right figure

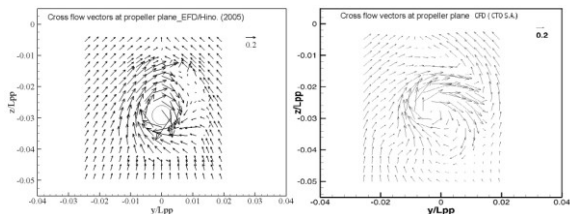


Figure 7: Comparison of cross flow vectors downstream of propeller plane ($x/LPP=0.9911$) EFD –left figure, CFD –right figure.

The comparison of the transverse distribution of all three velocity components downstream of plane of propeller is presented in Fig.8.

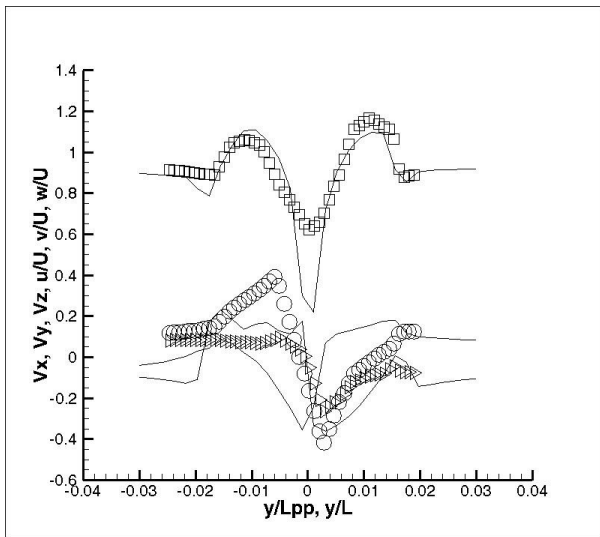


Figure 8: Velocity downstream of propeller plane ($x/LPP=0.9911$) at $z/LPP=-0.03$ (u/U : EFD open squares, v/U : EFD open triangles, w/U : EFD open circles, CFD - solid line.)

The pressure distribution on the hull surface is presented in Fig. 9.

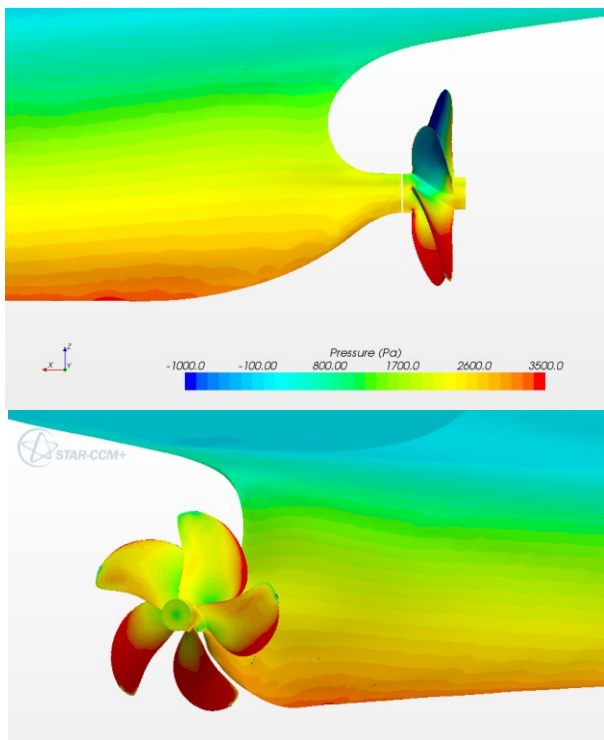


Figure 9: Hull surface pressure contours (port side and perspective view).

The results of KCS self-propulsion test simulation were presented and discussed at Gothenburg 2010 – A Workshop on Numerical Ship Hydrodynamics (Bugalski & Hoffmann 2010).

4 RESULTS OF NAWIGATOR XXI NUMERICAL SIMULATIONS

The main goal of the present project is to simulate the flow about complete ship with hull, rudder and propeller including the free surface around the ship. The simulation with the propeller behind the ship must to be run in transient mode, i.e. time accurate.

In order to verify the reliability of the CFD simulations, the flow about model of training vessel Navigator XXI trusted by CP469 propeller was computed and the results were compared with the existing experimental results (Bugalski & Kraskowski 2006). The computation results are presented in following figures.

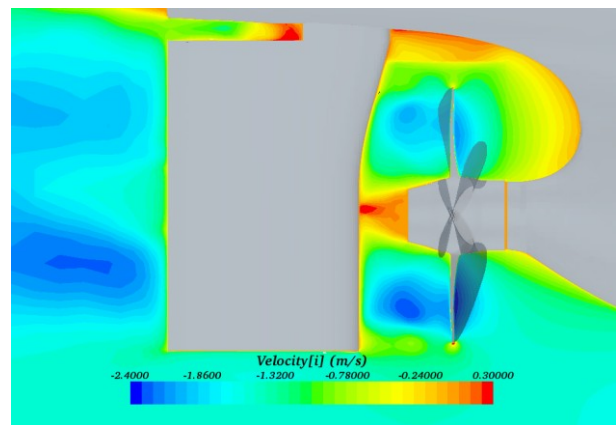


Figure 10: Axial velocity field in symmetry plane for Navigator XXI, CP469 propeller and R588 rudder (model scale).

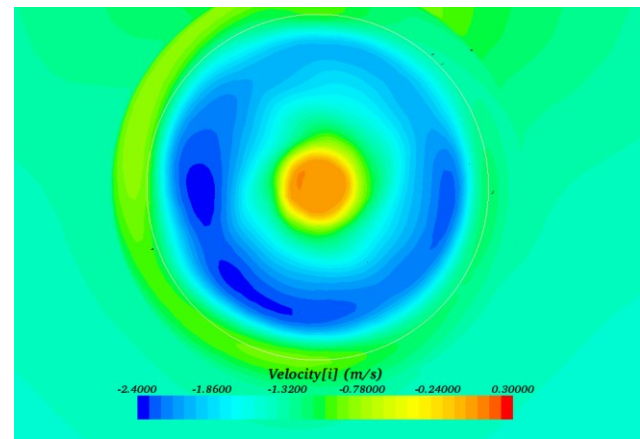


Figure 11: Axial component of total velocity field –in front of working propeller (Navigator XXI, CP469 propeller).

The study of the field quantities, velocity and pressure in the stern region shows a time varying, periodic flow field, which is related to the blade frequency of the rotating propeller. Figures 10, 11 and 12 show an example of the pressure field in the stern region and the axial velocity contours in a cross section at the rudder position. With respect to the velocity field, it shows that the propeller accelerates the flow and introduces swirl in the flow downstream of the propeller.

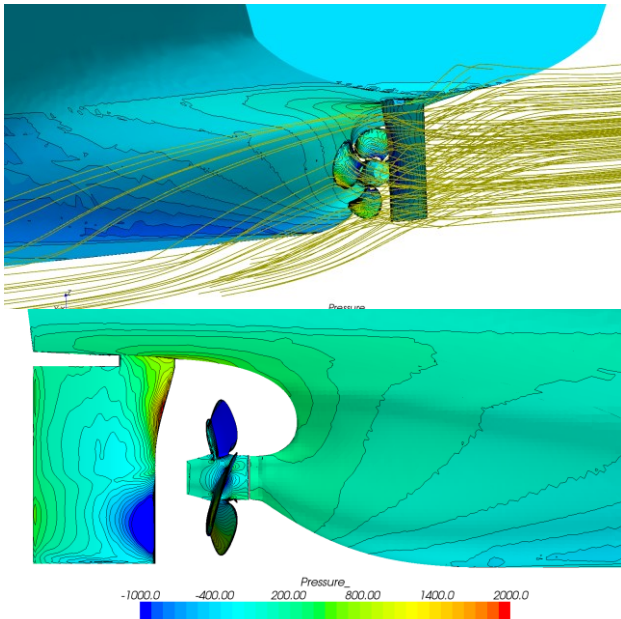


Figure 12: Pressure distribution on the stern of hull, propeller and rudder (perspective and starboard view)

As can be seen, the wake of the ship strongly influences the propeller, but, the propeller also influences the ship flow. This influence is most clearly seen in the pressure field. Upstream of the propeller the hull experiences suction, which reduces the pressure and increases the resistance. The effect is usually expressed as the thrust deduction. Another region of the hull that feels the presence of the propeller is the region above the propeller. In this region the passing blades will introduce pressure pulses on the hull, which, in critical cases, may lead to noise and the ship structure vibration.

5 RESULTS, DISCUSSION AND CONCLUSIONS

The planned results of research including development of discretization method for rotating propeller using STAR-CCM+ solver, development of computation method for hull-propeller system, and obtaining reliable results of the calculation of flow around a rotating propeller and viscous flow around a model ship with working propeller have all been achieved. Computations of flow around the ship hull with rotating propeller simulating propulsion tests have an unsteady character and therefore need more time to obtain convergent result. Calculations using sliding grids as in hull-propeller system are characterized by lower stability compared to simple flow computations, hence, using lower relaxation factors and higher number of iterations per time step is necessary. As a result, the calculation time increases significantly and the convergent result is more costly to obtain. At the same time the opportunity arises to analyze the phenomena occurring temporarily in the time domain. For the method to be practically useful, the accuracy of the calculated suction coefficient should be close to accuracy of the resistance determination. At the present stage of development the method is sufficiently accurate in prediction of the ship hull resistance and of the

propeller open water characteristics. However, the interaction phenomena are not predicted accurately enough. Further development work is planned in this area, involving modification of the grid structure, increasing grid density and application of other turbulence models.

ACKNOWLEDGEMENT

This research was supported by the Polish National Centre for Research and Development (NCBiR), grant No. R10 0039 06/2009) and Ministry of Sciences and Higher Education (MNiSW), grant No. A3:800-14/2009.

REFERENCES

- Bugalski, T. & Jaworski, S. (1984). 'Computer analysis of the interaction between the propeller and an axisymmetrical body'. XIII Seminar on Ship Hydrodynamics, Varna, Bulgaria.
- Bugalski, T. (1997). 'Modern Methods for Investigation of Hull -Propeller Interaction Phenomena using CFD'. NAV&HSMV International Conference, Sorrento, Italy.
- Bugalski, T. & Szantyr, J. (1998). 'Application of CFD for Analysis of the Ship and Propeller Flow'. TASK Quarterly Scientific Bulletin of the Academic Computer Centre 2(98).
- Bugalski, T. & Kraskowski, M. (2006). 'Validation of the RANSE wake computations for the training ship Navigator XXI and the dredger Uilenspiegel'. Proceedings of 9th Numerical Towing Tank Symposium (NuTTS), Le Croisic, France.
- Bugalski, T. & Hoffmann, P. (2010). 'Numerical Simulation of the Interaction between Ship Hull and Rotating Propeller'. Proceedings of Workshop on Numerical Ship Hydrodynamics Gothenburg 2010, Gothenburg, Sweden.
- Chao, K. (2005). 'Numeric Propulsion Simulation for the KCS Container Ship'. Proceedings of CFD Workshop Tokyo, Tokyo, Japan.
- Carrica, P., Fu, H. & Stern, F. (2010). 'Self-Propulsion Free to Sink and Trim and Pitch and Heave in Head Waves of a KCS Model'. Proceedings of Workshop on Numerical Ship Hydrodynamics Gothenburg 2010, Gothenburg, Sweden.
- Greve, M., Maksoud, M.-A., Eder, S. & De Causmaecker, J. (2010). 'Unsteady Viscous Flow Calculation around the KCS-model With and Without Propeller under Consideration of the Free Surface'. Proceedings of Workshop on Numerical Ship Hydrodynamics Gothenburg 2010, Gothenburg, Sweden.
- Hino, T. (ed.) (2005). Proceedings of CFD Workshop Tokyo 2005. NMRI Report 2005, Tokyo, Japan.
- Kim, W. J., Van, D. H. & Kim, D. H. (2001). 'Measurement of flows around modern commercial ship models'. Exp. In Fluids 31, pp. 567-578.

Larsson, L., Stern, F. & Visonneau, M. (2010). Proceedings of Workshop on Numerical Ship Hydrodynamics Gothenburg 2010, Gothenburg, Sweden.

Menter, F. & Maksoud, M.-A. (2002). 'Practical numerical simulation of viscous flow around ships'. The Naval Architect pp. 36-40.

Simonsen, C. D. & Carstens, R. (2008). 'RANS Simulation of the Flow around a Ship Appended with Rudder, Ice Fins and Rotating Propeller'. Proceedings

of 11th Numerical Towing Tank Symposium (NuTTS), Brest, France.

Van, S. H., Kim, W. J., Yim, G. T., Kim, D. H. & Lee, C. J. (1998). 'Experimental Investigation of the Flow Characteristics around Practical Hull Forms'. Proceedings 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan.