Numerical evaluation of the wave pattern for fast ships with taking into account the dynamic trim and sinkage

Marek Kraskowski
Centrum Techniki Okrętowej S. A.
(Ship Research and Design Centre)

ABSTRACT

This paper presents results of the numerical evaluation of the fast container ship's wave pattern, based on the RANSE (Reynolds-Averaged Navier Stokes Equations) method with dynamic trim and sinkage taken into account. Evaluating the ship's running attitude is based on coupling the flow solver with solving the motion equations for the ship hull. The results are presented for four speed values and contain: ship's running attitude (defined by changes of ship draught and trim angle), wave contours and wave profiles in chosen planes. The computed ship's running attitude and wave profiles are compared with the experimental results.

Keywords : RANSE, free surface, fast vessels, dynamic trim and sinkage

INTRODUCTION

Numerical tools enabling evaluation of the wave pattern are especially useful during initial optimization of the hull shape with respect to the wave-making resistance. The most important advantage of CFD (Computational Fluid Dynamics) methods is their low cost compared to the towing tank experiments. However to this time CFD calculations have been carried out mostly for fixed hull conditions, i.e. when dynamic trim and sinkage were neglected. Such approach is quite reasonable for low speeds when changes of trim and draught are so small that they have no noticeable influence on wave pattern, but it should not be used in the case of fast ships when the dynamic lift becomes significant.

Different approaches can be used for evaluating the running attitude of the ship, e.g. one can calculate it in an iterative manner by performing successive computations for fixed hull conditions and changing the position of the hull in each iteration basing on the computed forces and hydrostatic characteristics.

The presented method is based on coupling the RANSE solver for unsteady problem with solving the motion equations for the hull. Because exact history of hull motion is here of no importance, an artificial damping of the motion was used to improve stability of the procedure. The computations were carried out with the use of COMET flow solver extended with the user-programmed procedure for solving the motion equations.

PRINCIPLES OF THE FLOW SOLVING METHOD

The idea of RANSE approach is to decompose the variables in Navier-Stokes equations into the mean (time-averaged) and fluctuating component. The velocity components u_i are thus decomposed as follows :

$$u_i = \overline{u}_i + u_i'$$

where $\overline{u}_i$ is the mean velocity component and u_i' is the fluctuating component.

Likewise, pressure and other scalars are decomposed as follows :

$$\Phi = \overline{\Phi} + \Phi'$$

By substituting these expressions into the continuity and momentum conservation equations the following equations (for incompressible flow) are yielded :

continuity equation :

$$\frac{\partial}{\partial x_i}(\rho \overline{u}_i) = 0$$

momentum equation :

$$\frac{\partial}{\partial t}(\rho \overline{u}_i) + \frac{\partial}{\partial x_j}(\rho \overline{u}_i \overline{u}_j) =$$

$$= -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}(-\rho \overline{u_i u_j})$$

Terms $(-\rho \overline{u_i u_j})$ are called Reynolds stresses which are additional unknown variables, hence additional equations are required to close the system of equations. The equations are called the turbulence model. In the presented case, the $(k - \varepsilon)$ turbulence model was used, in which two equations are solved, namely the transport equations of two turbulence parameters : its kinetic energy k and rate of dissipation ε [3].

The RANSE approach can be shortly characterized as follows :

- the continuity and momentum conservation equations are solved for the averaged flow
- the turbulent flow is not calculated exactly. Instead, the turbulence is taken into account by solving the transport equations

for some statistic parameters of turbulence (in the presented case: turbulence kinetic energy and its rate of dissipation).

The numerical method used in this case for solving the partial differential equations is the Finite Volume Method. The idea of this method is to divide the considered flow domain into a finite number of control volumes (mesh generation) and to formulate equations in the integral form for each of the control volumes. The conservation equation for the general scalar quantity Φ in the integral form is expressed as follows :

$$\int_S \rho \Phi \mathrm{V} \mathrm{n} dS = \int_S \Gamma \operatorname{grad} \Phi \mathrm{n} dS + \int_\Omega q_\Phi d\Omega$$

where :

S – surface area which bounds the control volume Ω
Φ – scalar field function
V – velocity vector
n – vector normal to control volume surface
Γ – diffusivity
q_Φ – source of the quantity Φ

For each of the control volumes such equation is transformed into the algebraic equation by means of the discretization process, thus one obtains the system of algebraic equations which are to be solved in the iterative manner.

Because the flow around moving body is unsteady, the equation must be also discretized respective to time with a finite time step value.

The reason for using the Reynolds averaging instead of solving the exact Navier-Stokes equations is that the numerical method for exact equations would require:

- the size of control volumes comparable with the size of the smallest vortices in turbulent flow
- the time step value appropriate to resolve the unsteady phenomena of the turbulence.

For high Reynolds number (turbulent flow), to solve such problem is not possible because the necessary computational effort exceeds the possibilities of today's computers.

In the RANSE approach the size of control volumes and the time step should be appropriate to resolve the mean flow only. This greatly reduces the computational effort and still gives valuable results.

METHOD FOR EVALUATING THE FREE SURFACE

The COMET solver used for the flow computations offers two methods for evaluating the free surface :

- *interface tracking method* : the mesh of control volumes is deformed iteratively so as to satisfy the boundary conditions for free surface
- *volume – of – fluid method (VOF)* : an additional equation is solved for the scalar quantity determining the volume fraction of water in each point. In this method the computational domain contains both air and water.

The idea of both methods is presented in the sketches below.

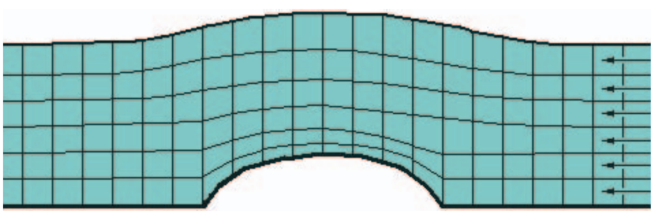


Fig. 1. *Interface tracking method : the mesh is deformed to satisfy the boundary conditions for free surface .*

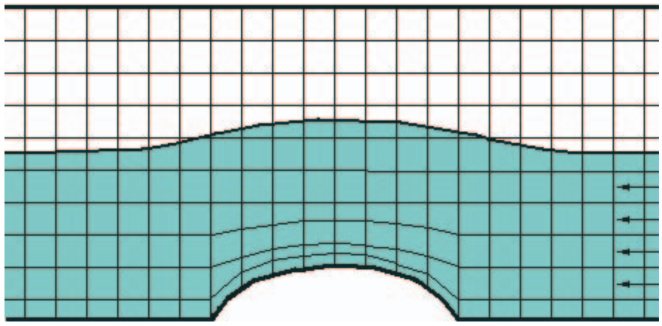


Fig. 2. *Volume – of – fluid method (VOF) : the mesh is fixed, an additional equation for the fluid transport is solved .*

The VOF method was chosen for the computations due to the following advantages of the method :

- There are no problems with complex geometries of the hull and such effects as wave breaking and air trapping which can occur in the flow around ship hull. In such case use of the interface tracking method would cause unacceptable distortion of the mesh cells
- The VOF method is more flexible when the dynamic mesh is applied, which is the case in the presented computations.

The VOF method is based on the following assumptions :

The fluids filling the domain are treated as one fluid whose properties depend on :

- physical properties of particular fluids
- local value of the so-called volume fraction C of particular fluids. The volume fraction C_i for i-th fluid varies from 0 to 1, where „0" means no i-th fluid in a given point, „1" means that only i-th fluid is present, values between 0 and 1 mean that the interface between the fluids occurs.

The properties of the effective fluid filling the domain are expressed as follows :

$$\rho = C_{air}\rho_{air} + C_{water}\rho_{water} \;\; ; \;\; \mu = C_{air}\mu_{air} + C_{water}\mu_{water}$$

where :

$$C_{air} + C_{water} = 1$$

There is no mixing between the fluids.

Additional equations for C are solved.

The mesh geometry is fixed.

The example of the free surface computed with the use of VOF method for the ship model, compared with the experimental one, is shown in Fig.3.

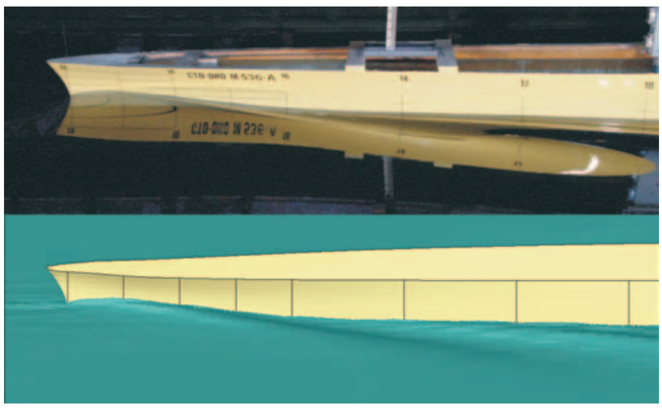


Fig. 3. *Comparison of the experimental and computed free surface for the ship model (experiment - by P. Grzybowski, computation - by M. Kraskowski) .*

METHOD FOR COUPLING THE FLOW SOLVER WITH SOLVING THE MOTION EQUATIONS

Let us introduce two coordinate systems :

→ the global coordinate system XYZ, moving in the direction of ship motion with the same speed, and
→ the local coordinate system xyz connected with the ship's centre of gravity, moving and rotating together with it.

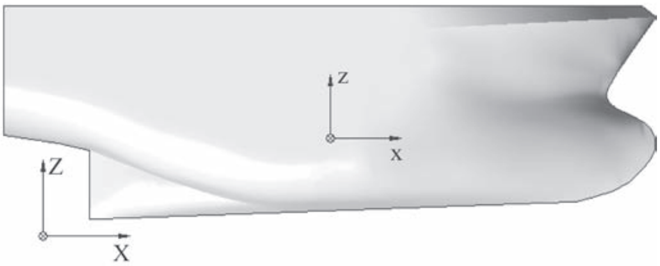


Fig. 4. *Coordinate system .*

The basic idea of the presented method for wave pattern computations with dynamic trim and sinkage taken into account is that the hull motion directly depends on the forces acting on it and the mesh of control volumes is moving together with the moving hull without relative motion between the mesh nodes. The flow is solved in the global coordinate system and the motion of the control volumes is taken into account in the motion equations. The boundary conditions are also given in the global coordinate system.

The flow solver is programmed to calculate the vertical force and trimming moment acting on the hull (in this case the moment vector points in „y" direction). On the basis of the forces the translational and angular accelerations of the hull are computed in each time step. The accelerations are used to update the translational and angular velocity in each time step and these are used to update the vertical position and trim angle. As mentioned before, the exact time history of motion is of no importance since we are interested in the steady state solution, hence some modifications of the equations can be applied to improve stability of the method.

Therefore the exact algorithm used to evaluate the ship's running attitude can be described as follows :

★ Calculate the vertical force F_z and trimming moment M_Y in the time step t_n

★ Calculate values of the translational acceleration a_z and rotational one ε_y in the time step t_n , introducing the artificial damping proportional to velocity values :

$$a_Z = \frac{F_Z - mg}{m} - \alpha V_Z \quad ; \quad \varepsilon_Y = \frac{M_Y}{I} - \beta\omega_Y$$

where :

m	-	hull mass
I	-	moment of inertia
g	-	acceleration of gravity
V_Z	-	translational velocity
ω_Y	-	rotational velocity
α and β	-	proportionality factors of positive value.

★ Use the acceleration values computed in the previous time step t_{n-1} to compute the average values and use them as values for the current time step:

$$a_Z^n = 0.5(a_Z^n + a_Z^{n-1}) \quad ; \quad \varepsilon_y^n = 0.5(\varepsilon_Y^n + \varepsilon_Y^{n-1})$$

★ Update the values of the translational velocity V_z and angular velocity ω_Y :

$$V_Z^n = V_Z^{n-1} + a_Z^n \cdot \Delta t \cdot D_V \quad ; \quad \omega_Y^n = \omega_Y^{n-1} + \omega_Y^n \cdot \Delta t \cdot D_\omega$$

where :

D_V and D_ω - so-called delay factors for the translational and angular velocity, respectively. Values of the delay factors are contained between 0 and 1.

★ Update the vertical position and trim angle :

$$Z^n = Z^{n-1} + V_Z^n \cdot \Delta t \quad ; \quad \varphi^n = \varphi^{n-1} + \omega_Y^n \cdot \Delta t$$

★ Go to the next time step.

The modifications of the equations, which improve the stability but make the motion history not exact, are the following : artificial damping and delay of motion. The algorithm in question can be adjusted to simulate the dynamics simply by setting the factors α, β, D_V and D_ω to zero.

DESCRIPTION OF THE TEST CASE

The method was tested for the fast containership hull of the parameters presented in the table below.

Length b.p.	L	135.25	[m]
Breadth	B	8.45	[m]
Draught	T	4.14	[m]
Block coefficient	C_B	0.442	[-]
Prismatic coefficient	C_P	0.658	[-]
Waterline coefficient	C_W	0.748	[-]

The below given sketch shows the body lines of the hull.

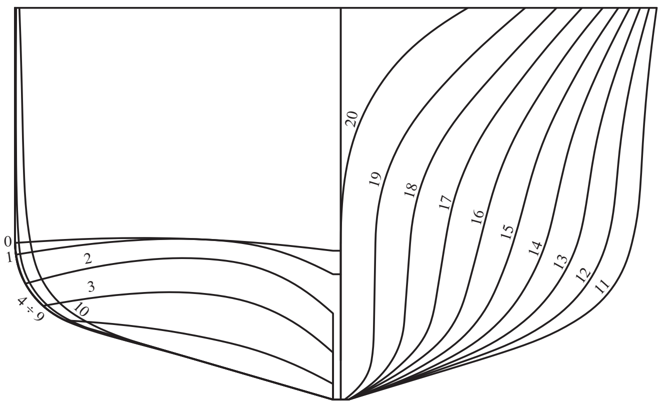


Fig. 5. *Body lines of the ship hull used for the test.*

The computations were performed for the ship in model scale in order to make direct comparison with the experiment possible. The model scale was equal to [1 : 20.92], hence the model length was 6.47 m.

Four values of model speed were considered. The values are listed in the below given table together with the corresponding values of ship speed and Froude number.

No.	Model speed [m/s]	Ship speed [m/s]	Froude number
1	2.389	10.928	0.3
2	3.186	14.570	0.4
3	3.982	18.213	0.5
4	4.778	21.855	0.6

MESH GENERATION

Generating the mesh of control volumes is the largest work that the user of CFD software has to do when performing the flow computations. Of course, this process is computer-aided but to obtain a good mesh some knowledge and experience is always required. The basic requirements for the mesh used to solve the ship flow are as follows :

- The size of the flow domain divided into the mesh of control volumes should be large enough to avoid the effects of restricted water
- If the flow is symmetric, which is the case here, only one half of the hull should be considered with appropriate boundary conditions on the symmetry plane
- In regions where the flow variables change rapidly, the size of control volumes (cells) should be small and the size of adjacent cells should be comparable. This is particularly important in the near-wall region where the velocity gradient is high, and in the free-surface region where the volume fraction of water changes rapidly from 0 to 1. It is difficult to specify the required size of cells exactly hence the quality of the mesh should be verified after performing some initial computations. When plotting the flow variables one should obtain smooth contour lines everywhere (this is a very simple, but useful in practice, engineering criterion)
- In the regions located far from the region of interest the elements should be large to minimize the computational effort.

The COMET solver requires the mesh to be constructed of hexahedral cells for free-surface computations. Other types of mesh used in CFD are: tetrahedral and polyhedral (polyhedral are cells with arbitrary number of faces). Hexahedral mesh is always recommended to use whenever possible because the discretization of integral equations on hexahedral cells is very natural and such mesh offers good stability and quality of results.

The mesh for the presented computations was generated with the use of ICEM Hexa mesh generator. To generate the mesh the user has to do the following tasks :

- To define the surfaces bounding the flow domain. The below given sketch shows the edges of the computational domain – the hull is placed in the rectangular block.

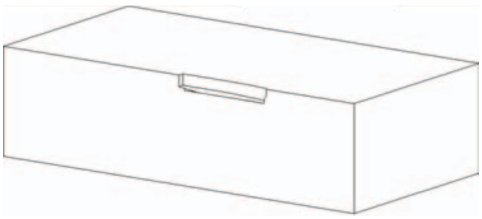


Fig. 6. Edges of the flow domain .

- To divide the domain space into hexahedral blocks. This task is the most difficult because there is no algorithm to do it and the mesh quality depends mainly on the quality of blocks. Fig.7 shows the edges of the blocks.

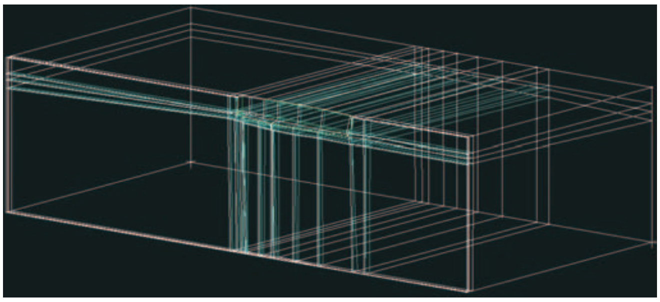


Fig. 7. Flow domain divided into blocks .

- To divide the blocks into cells. It is simply done by dividing the edges of the blocks. Fig.8 through 10 show the ready mesh.

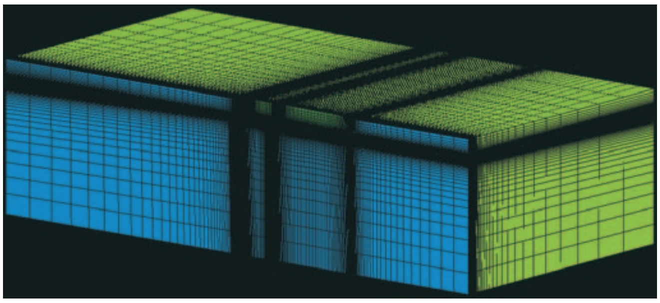


Fig. 8. Mesh of control volumes for entire domain .

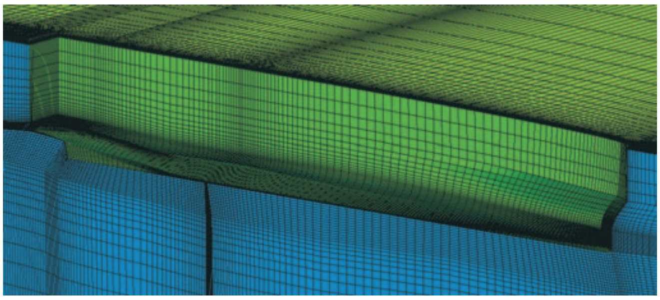


Fig. 9. Mesh of control volumes for hull region .

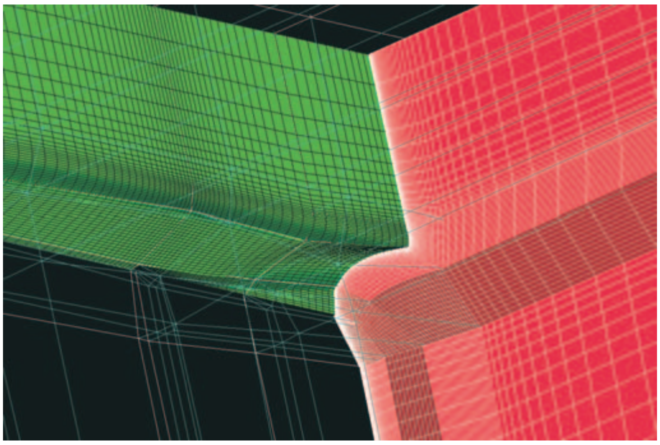


Fig. 10. Mesh details : mesh on the hull surface, section of the mesh interior, edges of the blocks .

GENERAL PARAMETERS AND PROCEDURE OF THE COMPUTATIONS

The unsteady computations with the use of RANSE method are performed in an iterative manner, and the following levels of iterations can be distinguished :

- The solution is step-by step advanced with time
- Iterations are performed to compute the flow for the current time step, i.e. to satisfy the conservation equations (these are called the outer iterations)
- For each of the outer iterations, iterations are performed to solve the system of algebraic equations (these are called the inner iterations).

In the presented computations the following procedure was used to obtain their convergence :

- The uniform flow was taken as the initial condition
- The computation for the fixed model was carried out till the convergence of results for the forces acting on the hull were reached. During this computation one outer iteration

per time step was executed. This is the common way of obtaining the steady-state solution, called pseudo time--marching

- When the result convergence for the forces was achieved the motion of the hull was released. The number of outer iterations per time step was increased to 5 and the computation was continued till the result convergence for the hull position were reached.

The figures below show the example history of the motion after releasing the hull translation, translational acceleration and velocity for the Froude number Fn = 0.6. The acceleration shows the tendency to oscillate, nevertheless the oscillations do not significantly affect the smoothness of the motion.

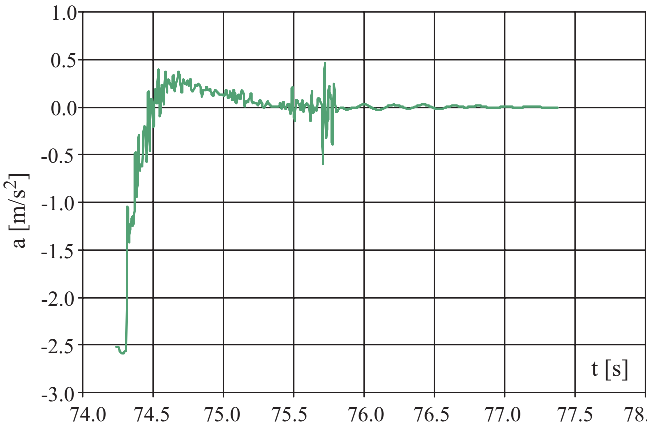


Fig. 11. *Time history of the translational acceleration .*

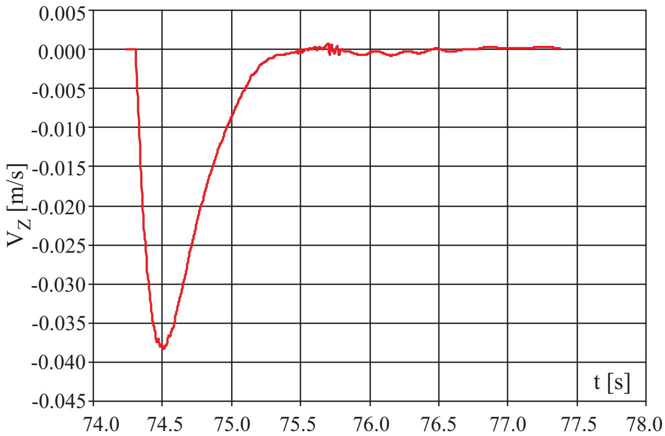


Fig. 12. *Time history of the translational velocity .*

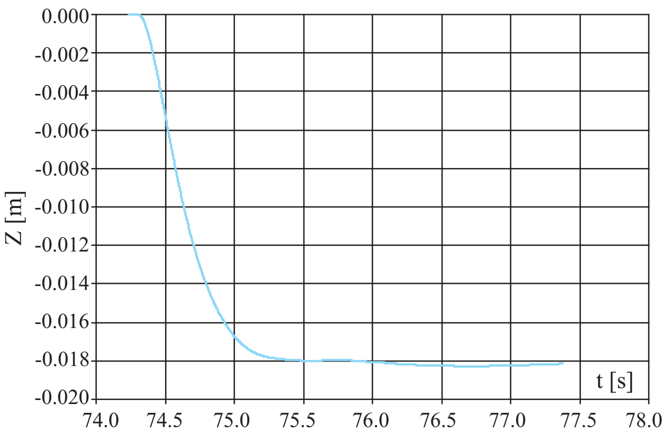


Fig. 13. *Time history of the total translation .*

RESULTS

The following results are presented :

- computed running attitude of the hull, defined by the change of the draught (sinkage) Z and trim angle φ (compared with the experimental values)
- wave contours
- wave profiles (compared with the experimental profiles).

Fig.14 and 15 show the computed and measured running attitude of the hull in function of Froude number.

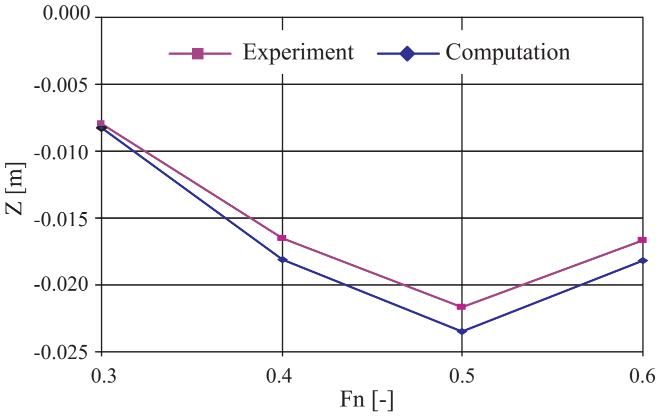


Fig. 14. *Computed and measured sinkage .*

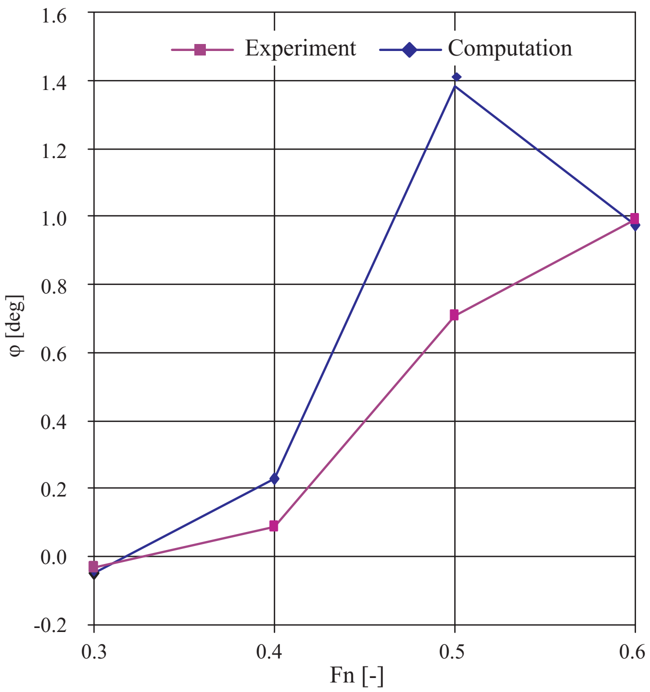


Fig. 15. *Computed and measured trim angle .*

Fig.16 through 19 show the wave contours corresponding to Froude number values : 0.3, 0.4, 0.5 and 0.6.

Tab.1 presents the wave profiles in the planes parallel to the symmetry plane, located at Y=B and Y=3B from the symmetry plane, where B is the ship breadth. The computed profiles are compared with those measured. Thick lines indicate the location of aft perpendicular (at X = 0 m) and fore perpendicular (at X = 6.47 m).

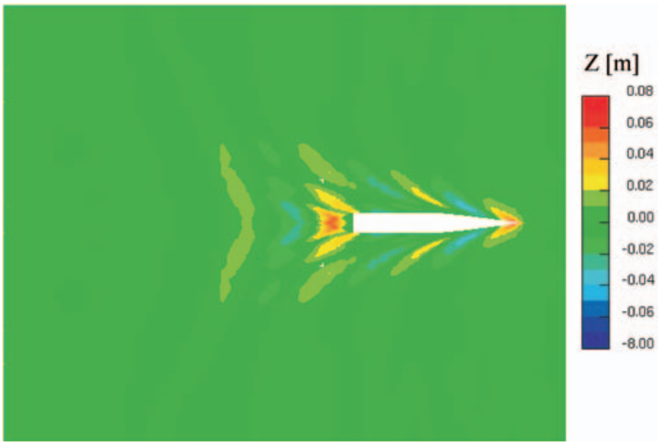


Fig. 16. Wave contour at Froude number Fn = 0.3 .

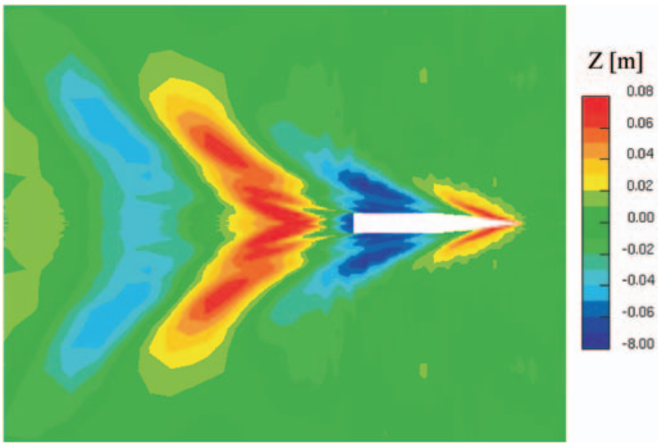


Fig. 18. Wave contour at Froude number Fn = 0.5.

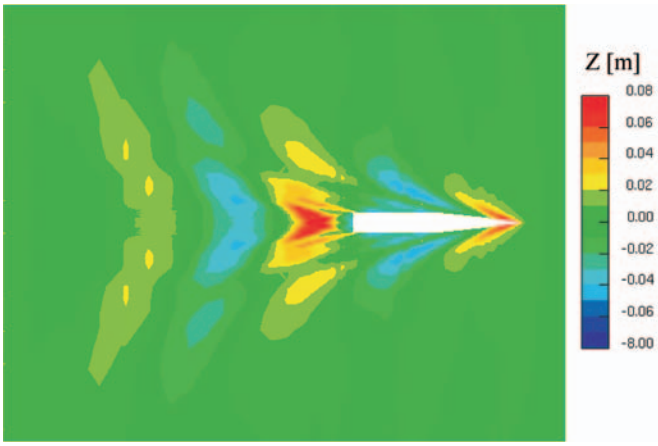


Fig. 17. Wave contour at Froude number Fn = 0.4.

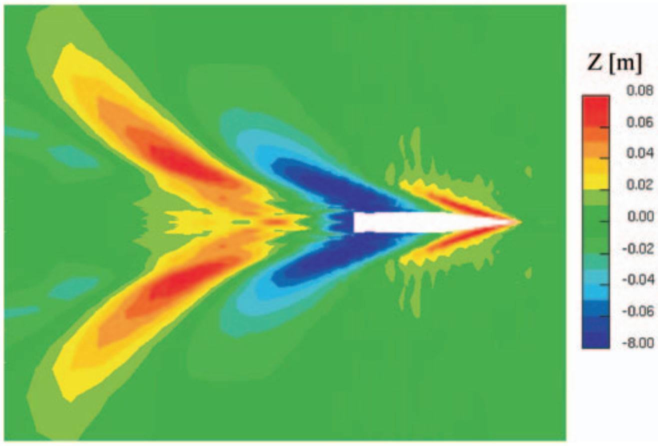
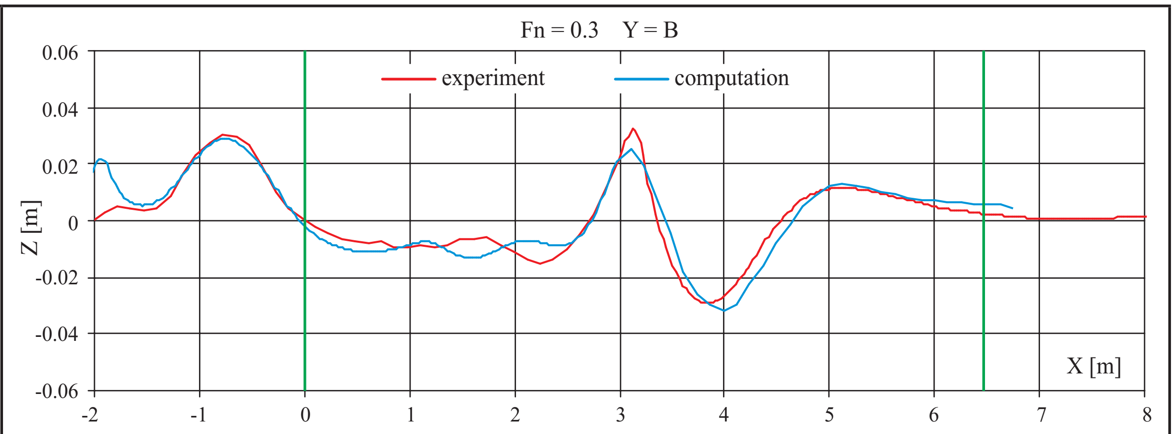
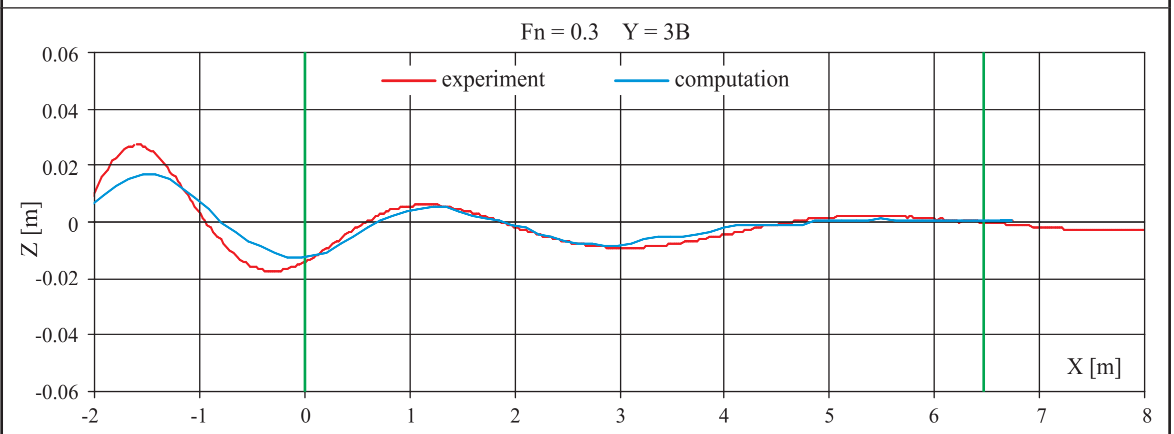
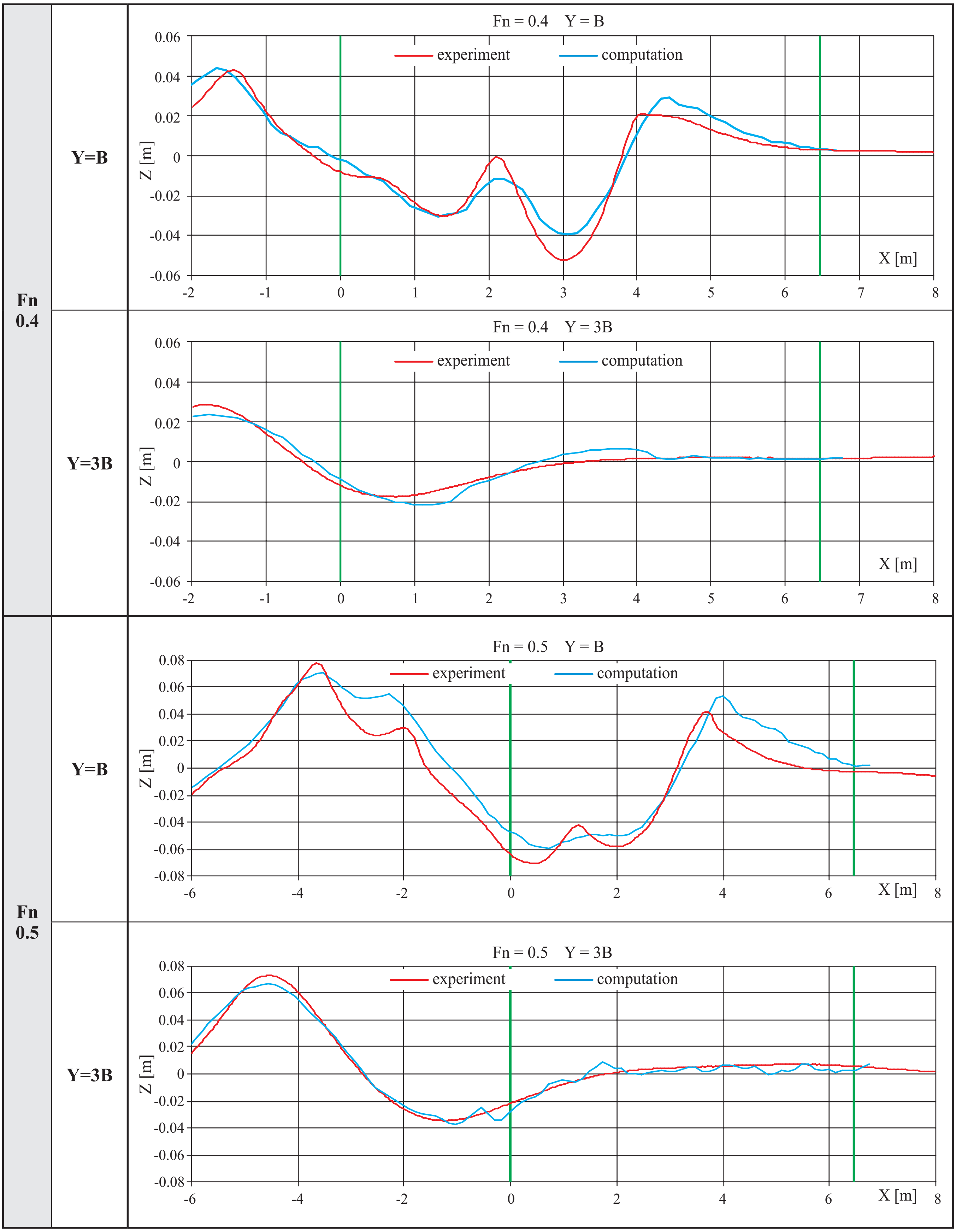
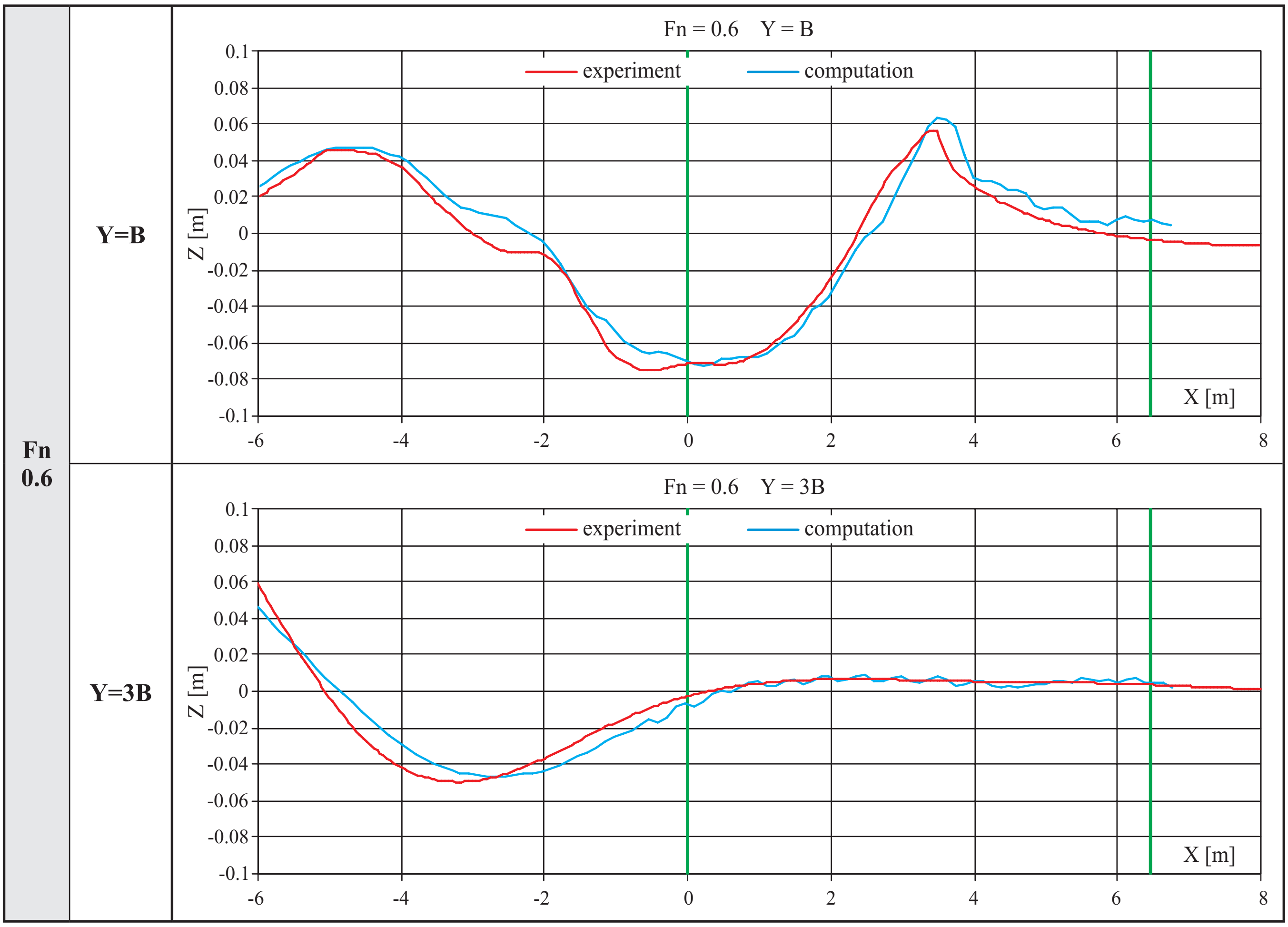


Fig. 19. Wave contour at Froude number Fn = 0.6.

Tab. 1. Wave profiles in the planes parallel to the symmetry plane

Fn 0.3	**Y=B**	
	Y=3B	


Fn 0.4
Y=B
Fn = 0.4 Y = B
experiment
computation
Z [m]
X [m]
Y=3B
Fn = 0.4 Y = 3B
experiment
computation
Z [m]
X [m]
Fn 0.5
Y=B
Fn = 0.5 Y = B
experiment
computation
Z [m]
X [m]
Y=3B
Fn = 0.5 Y = 3B
experiment
computation
Z [m]
X [m]



CONCLUSIONS

The comparison of the obtained results with the experiment yields the following conclusions :

- The best agreement of the computed and measured ship's running attitude as well as the wave profiles was obtained for extreme values of Froude number considered in the computations, i.e. Fn = 0.3 and Fn = 0.6
- The largest error of the computed running attitude occurred for Froude number Fn = 0. 5. At this value the maximum computed trim angle is observed, while in the experiment the trim angle increases monotonically when the speed increases. The Froude number range between 0.3 and 0.6 seems the most „difficult" with respect to the numerical prediction of the flow since in this range the transition between displacement floating and planing occurs. Nevertheless the predicted wave profiles for Fn = 0.4 and Fn =0.5 are also in a good agreement with the experiment.
- For Fn = 0.5 and Fn =0.6 some short, non-physical waves are observed near the bow part of the hull, which results from the mesh geometry influence.

The final conclusion is that :

- The proposed method for evaluating the wave pattern with accounting for the dynamic trim and sinkage is robust and accurate enough to estimate the hull quality with respect to the generated wave pattern. It is worth of further validation and development in order to improve the quality of the results.
- The successful attempt on coupling the flow solver with the body motion computations encourages developing the body motion module and using it to simulate dynamic phenomena, e.g. launching. Further challenge is the simulation of hull motion in waves.

Acknowledgement

The research presented in this paper has been financially supported by the Polish Ministry of Scientific Research and Information Technology (Grant No. 5T12C 074 24). The author would like to express his gratitude for this support.

NOMENCLATURE

a_Z - acceleration in „Z" direction
C_i - volume fraction of the i-th fluid
D_V - translational velocity delay factor
D_ω - angular velocity delay factor
Fn - Froude number
F_Z - vertical force
g - acceleration of gravity
I - hull moment of inertia
m - hull mass
M_Y - trimming moment
n - vector normal to control volume surface
p - pressure
$\overline{p}$ - time-averaged pressure
q_Φ - source of the quantity Φ
S - surface bounding the control volume
t - time
u_i , u_j - velocity components (in Cartesian coordinate system)
$\overline{u}_i$, $\overline{u}_j$ - time-averaged velocity components

u'_i - velocity fluctuation component

V - velocity vector
V_Z - translational velocity
x_i , x_j - components of location vector
Z - translation in „Z" direction

α - translational motion damping factor
β - angular motion damping factor
Γ - diffusivity
Δt - time step
ε_Y - angular acceleration
μ - dynamic viscosity
ρ - density
φ - trim angle
Φ - general scalar quantity
$\overline{\Phi}$ - averaged value of general scalar quantity
Φ' - fluctuation of general scalar quantity
ω_Y - angular velocity
Ω - control volume

BIBLIOGRAPHY

1. Azcueta R.: *Computation of Turbulent Free-Surface Flows Around Ships and Floating Bodies*". PhD thesis. Report No. 612 AB 3-13 TUHH
2. Ferziger J.H., Peric M.: *Computational Methods for Fluid Dynamics*, Springer. Berlin, 1999
3. Wilcox D.C.: *Turbulence Modeling for CFD*, DCW Industries. 2002

CONTACT WITH THE AUTHOR

Marek Kraskowski, M.Sc., Eng.
Ship Hydromechanics Division,
Research and Development Department,
Ship Design and Research Centre – Stock Company
Szczecińska 65
80-392 Gdańsk, POLAND
e-mail : Marek.Kraskowski@cto.gda.pl

Miscellanea

Scientific meeting

On 5 December 2005 the plenary meeting of the Marine Technology Unit (acting in the frame of the Transport Technical Means Section, Transport Committee, Polish Academy of Sciences), was held at the Faculty of Maritime Technology (WTM), Szczecin University of Technology.

Two papers prepared by WTM scientific workers, were presented :

★ *Model tests of ship fluidal boilers* – by W. Zeńczak
★ *Starting the shipboard high-power devices on ships equipped with central hydraulic supply system* – by A. Banaszek

An interesting discussion on both the papers was held after the presentation.

Next, the Unit's members discussed organizational problems concerning its current activity and working plan for the year 2006.

Conference

KONES 2005

On 4-7 September 2005 already
31st International Scientific Conference on :

Internal Combustion Engines

took place at Polanica Zdrój, a health resort at the foot of the Stołowe Mountains in south-west Poland. Institute of Aeronautics, Wrocław University of Technology, and Polish Academy of Sciences were the hosts of the Conference. Its program of a very wide range of topics contained presentation of 96 papers including 6 plenary ones, namely :

❖ *Ignition control in the HCCI (Homogenous Charge Compression Ignition) combustion engine system fuelled with methanol-reformed gases* – by Toshio Shudo (Hokkaido University, Japan)

❖ *Development of a 125cc two- stroke step-piston engine using a one-dimensional engine code* – by A. A. Aziz, Z. A. Latif, M. F. M. Mohamad, G. L. Ming (University Teknologi, Malaysia)

❖ *HCCI with selected standard and alternative fuels : challenges and solutions* – by M. L. Wyszyński (The University of Birmingham, UK) and H. Xu (Jaguar Cars, Coventry, UK)

❖ *Development of a DME (dimethyl ether) fueled heavy-duty engine with lean NOx trap* – by Yoshio Sato (National Traffic Safety and Environment Laboratory, Japan) and Takayuki Tsuchiya (Nissan Diesel Motor Co Ltd, Japan)

❖ *OSD clean fuel initiative* – by A. Sandel (US Army RDECOM, USA)

❖ *Limits of internal combustion engines efficiency* by J. Macek (Czech Technical University in Prague)

54 papers presented during panel sessions were divided into two topical groups :

Ecology, Combustion, Thermodynamic Processes, Fuelling (30 papers)

Design, Operating, Measurement, Control (24 papers)

36 remaining papers were topics of a poster session.

It should be stressed that many universities and scientific research centres, both Polish and foreign ones, were interested in active participation in the Conference. Polish authors represented as many as 29 centres among which the greatest number of papers (12 papers each) was prepared by authors from Wrocław University of Technology and Cracow University of Technology. Whereas foreign authors who submitted 18 papers together, represented the scientific research centres from Czech Republic, Lithuania, Japan, Malayasia, Germany, Slovak Republic, Switzerland, United Kingdom and USA.