Vuko VUKČEVIĆ, Fakultet strojarstva i brodogradnje, Sveučilište u Zagrebu, Ivana Lučića 5, HR-10000 Zagreb, Croatia, vuko.vukcevic@fsb.hr
Hrvoje JASAK, Fakultet strojastva i brodogradnje, Sveučilište u Zagrebu, Ivana Lučića 5, HR-10000 Zagreb, Croatia, hrvoje.jasak@fsb.hr

OPENFOAM IN MARINE HYDRODYNAMICS

Abstract

The subject of this paper is the application of OpenFOAM software in marine hydrodynamics. Mathematical model for two phase flows using the Volume of Fluid method is briefly presented. Finite volume method is used for equation discretization. The simulation results of steady state resistance in calm seas are compared with experimental data for two hull forms. Seakeeping simulations are also presented. Since OpenFOAM supports mesh motion, it can be used for forced oscillation simulations. As an example, results of harmonic sway motion with forward speed are presented. Finally, the seakeeping simulation of ship advancing in head waves with constant forward speed is presented.

Key words: OpenFOAM, finite volume method, steady state resistance, seakeeping

OPENFOAM U BRODSKOJ HIDRODINAMICI

Sažetak

Tema ovog rada je primjena programskog paketa OpenFOAM u brodskoj hidrodinamici. Matematički model dvofaznog strujanja sa slobodnom površinom uz korištenje "Volume of Fluid" metode je ukratko prikazan. Za diskretizaciju jednadžbi se koristi metoda kontrolnih volumena. Rezultati otpora broda na mirnom moru uspoređeni su s eksperimentalnim podatcima za dvije brodske forme. Prikazane su i simulacije vezane za pomorstvenost plovnih objekata. Razvijeni programski paket podržava pomične mreže, te se može koristiti za simulacije sa nametnutim harmonijskim gibanjem pojedinog stupnja slobode. Kao primjer, prikazani su rezultati harmonijskog zanošenja broda pri konstantoj brzini napredovanja. Konačno je prikazana simulacija poniranja i posrtanja broda koji napreduje konstantnom brzinom u polju nailaznih valova.

Ključne riječi: OpenFOAM, metoda kontrolnih volumena, otpor broda, pomorstvenost

1. Introduction

Growing computational resources have made Computational Fluid Dynamics (CFD) an active area of research over the past few decades. Marine hydrodynamics is a very important and broad part of that area. Marine hydrodynamic flows are incompressible, two-phase, turbulent and very often unsteady. General toolbox for simulation of such flows is OpenFOAM and its capabilities are briefly presented in this paper. OpenFOAM is versatile open source software package written in C++. It heavily relies on object oriented paradigm [1], making it easy to use and customize.

This paper is organized as follows. Section 2 presents mathematical modelling of governing physics. Section 3 briefly describes numerical implementation using Finite Volume Method (FVM). In following sections, three types of simulations are presented: steady state resistance in calm seas, forced oscillatotion motion and seakeeping simulation of a ship advancing in head waves. Finally, a short conclusion is drawn.

2. Governing equations

This section presents the mathematical model of incompressible, two-phase flow with Volume of Fluid (VOF) method for interface capturing.

2.1. Continuity and Navier – Stokes equations

Two-phase, incompressible and turbulent flow is often modelled with continuity equation (1) and Navier – Stokes equations (2):

$$\nabla \cdot \mathbf{U} = 0, \tag{1}$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U}\mathbf{U}) - \nabla \cdot \left(\mu_{eff} \nabla \mathbf{U}\right) = -\nabla p + \rho \mathbf{g} + \nabla \mathbf{U} \cdot \nabla \mu_{eff} + \sigma \kappa \nabla \alpha, \tag{2}$$

where $\mathbf{U}$ denotes velocity vector, p pressure, ρ and μ_{eff} are density and dynamic viscosity, respectively. $\mathbf{g}$ denotes gravitational acceleration. σ and κ present the surface tension coefficient and mean free surface curvature, respectively. α is the volume fraction variable which will be described in the following section. For more details on the derivation of the above equation, reader is referred to [2].

2.2. Volume of Fluid equation

Equations (1) and (2) should be solved for two phases (water and air) and then coupled at the free surface. To prevent such procedure, VOF equation is introduced and the fluid is modelled as one continuum of mixed properties. Density and viscosity fields can be written as:

$$\rho = \alpha \rho_1 + (1 - \alpha)\rho_2, \tag{3}$$

$$\mu = \alpha \mu_1 + (1 - \alpha)\mu_2. \tag{4}$$

In above equations, index 1 presents water properties while index 2 presents air properties. α has the value of 1 in the water and value of 0 in the air. Governing volume fraction equation has the following form:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) + \nabla \cdot (\alpha (1 - \alpha) \mathbf{U}^{\mathrm{r}}) = 0. \tag{5}$$

The first two terms depict usual advection equation while the last term is introduced for interface compression [3]. It is needed to maintain a sharp interface between two phases.

2.3. Turbulence modelling

In industrial applications, turbulence is most often modelled with eddy viscosity two-equation models [4]. Such models solve two additional partial differential equations with emphasized local character. Inclusion of such equations leads to a small computational overhead of around 2 to 5%. The $k - \omega\, SST$ [5] model is used exclusively in this paper. After the solution, once the turbulent kinetic energy k and specific dissipation ω are known, one can easily calculate μ_{eff} needed for equation (2).

3. Numerical procedure

Finite Volume Method (FVM) is used to discretize the governing equations presented in the previous section. Following procedures in [6], second order accuracy in space and time is achieved. Pressure-velocity coupling is obtained through segregated algorithm called PIMPLE, a combination of SIMPLE and PISO. It allows for higher Courant numbers and decreases overall computational time. Previous standard procedure was to solve the VOF equation explicitly; leading to a maximum Courant number of 1. Using bounded, second order numerical schemes, every term in the VOF equation is treated implicitly. Two applications are used in this paper: `steadyNavalFoam` which is the accelerated steady state solver, and `navalFoam` which is the transient solver that allows arbitrary variants of mesh motion, wave models, etc.

Wave modelling is obtained with relaxation zones [7]. Such zones are usually positioned near the inlet and outlet boundaries where the potential flow solution is smoothly blended with CFD solution. As opposed to [7], blending is achieved implicitly.

4. Steady state resistance

This section presents simulations regarding steady state resistance in calm water. Two hull forms are considered: KRISO Container Ship (KCS) [8] and US Navy Combatant DTMB 5415 [9]. KCS has no appendages except for rudder. DTMB hull is only appended with bilge keels.

4.1. DTMB 5415 with bilge keels

Perspective view of the hull geometry is presented in Fig 1. Geometry and test conditions are readily available in [9] and will not be described here. Tested speed is 2.241 m/s corresponding to the Froude number of 0.41. Mesh includes full model of the ship (without symmetry plane) and it consists of approximately 1 600 000 polyhedral cells. Fig 2. presents drag force convergence with `steadyNavalFoam` (black line). Experimental force is presented with a constant value (red line). Relative error in drag force when compared to experimental data is 4.9%. This result would be better if one would use a finer mesh, especially in regions near the free surface and generally near the hull. It should also be noted that the CPU time for this simulation is about 2 hours on 4 cores, Intel i7 processor, with 3.7GHz and 16GB of RAM.

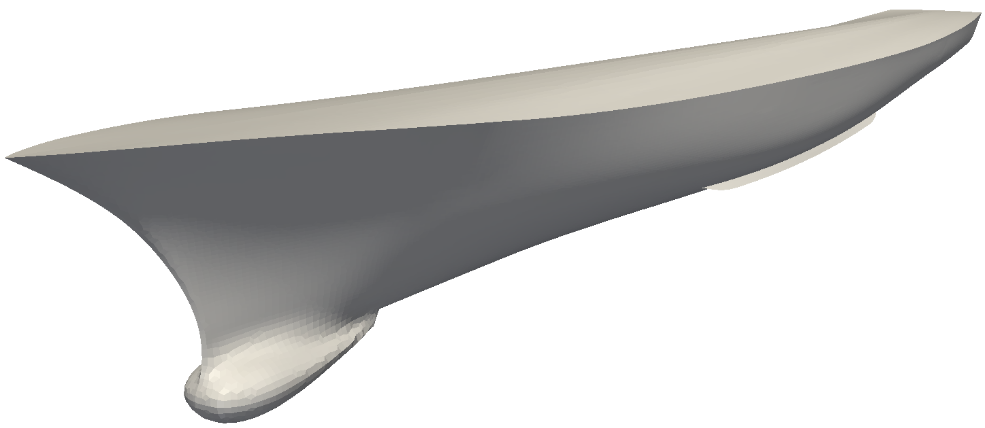

Fig 1. DTMB 5415 with bilge keels, perspective view

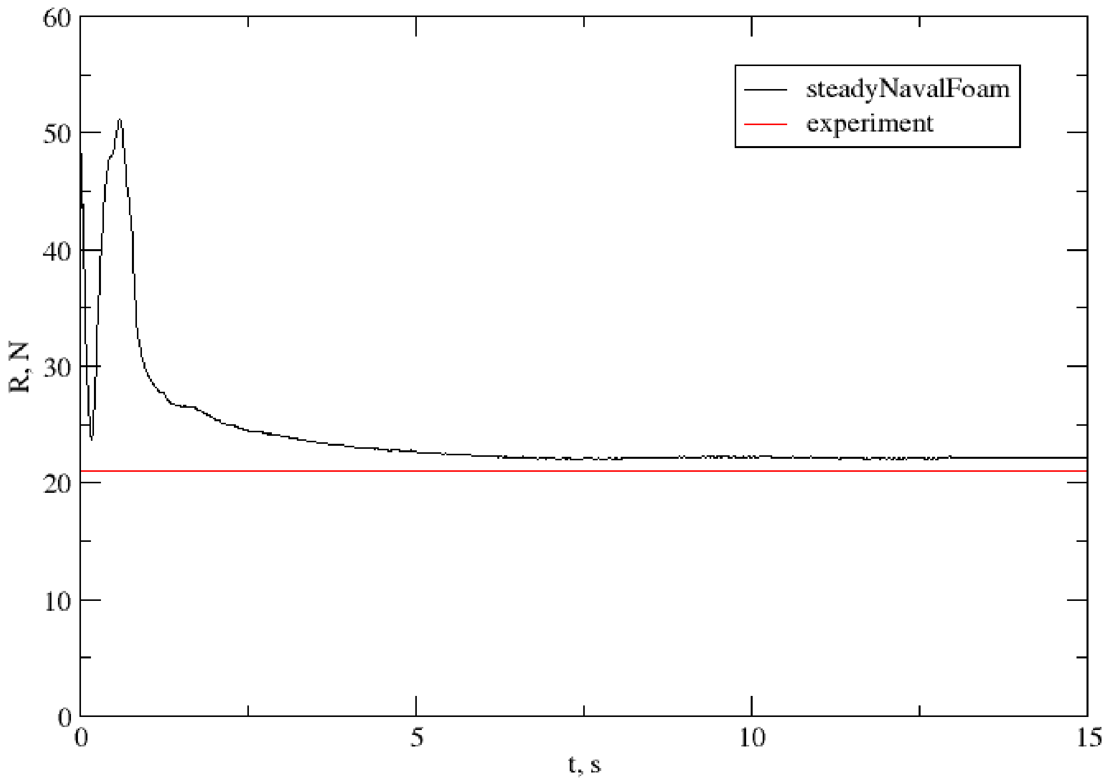

Fig 2. Drag force convergence and experimental comparison, DTMB 5415, $Fn = 0.41$

4.2. KCS with rudder

For KCS simulation, using symmetry plane boundary conditions allowed only half of the model to be considered. Geometry is shown in Fig 3. Tested speed is 2.196 m/s, which corresponds to the Froude number of 0.26. The mesh consists of approximately 950 000 cells and is carefully refined in areas of interest. Fig 4. presents drag force convergence and comparison with experimental results. In this case, the relative error in total drag force is 1.9%. It should be noted that the negative drag force is obtained because of the different orientation of coordinate system. Steady state solution is achieved after appoximately 70 s of simulation time, and the coressponding free surface elevation is shown in Fig 5 in meters. Blue and red colours show wave throughs and crests, respectively. CPU time for this simulation is less than 1 hour.

Fig 3. KCS hull with rudder, side view

5. Forced oscillation simulations

Transient simulations, such as forced oscillation simulations are carried out with a transient solver `navalFoam`. Its implicit nature and state of the art numerical procedures allow very high Courant numbers, leading to a shorter simulation. The mesh is coarse, consisting of 600 000 cells. Sway motion for a full scale KCS ship [8] with forward speed of 15 knots is considered. Frequency range is varied from 0.2 to 1.1 rad/s, with a step of 0.1 rad/s. Sway amplitude is set to 1 m for all simulations. Sway force amplitude is determined from the sway force signal after three initial, transient periods. Fig 6. presents sway force amplitudes as a function of angular frequencies (sway transfer function). Results are compared with a 3D Boundary Element Method (BEM) solution obtained with HYDROSTAR [10]. Free surface in meters is shown in Fig 7. after 4 oscillation periods. It can be seen that radiation waves have greater effect than waves coming from forward speed. Since the mesh is coarser in the far field, waves get damped away from the region of interest. Courant number used for all simulations was 500, leading to an average CPU time of about 5 hours (dependant on the frequency). Simulation time was executed for approximately 6 periods of oscillation.

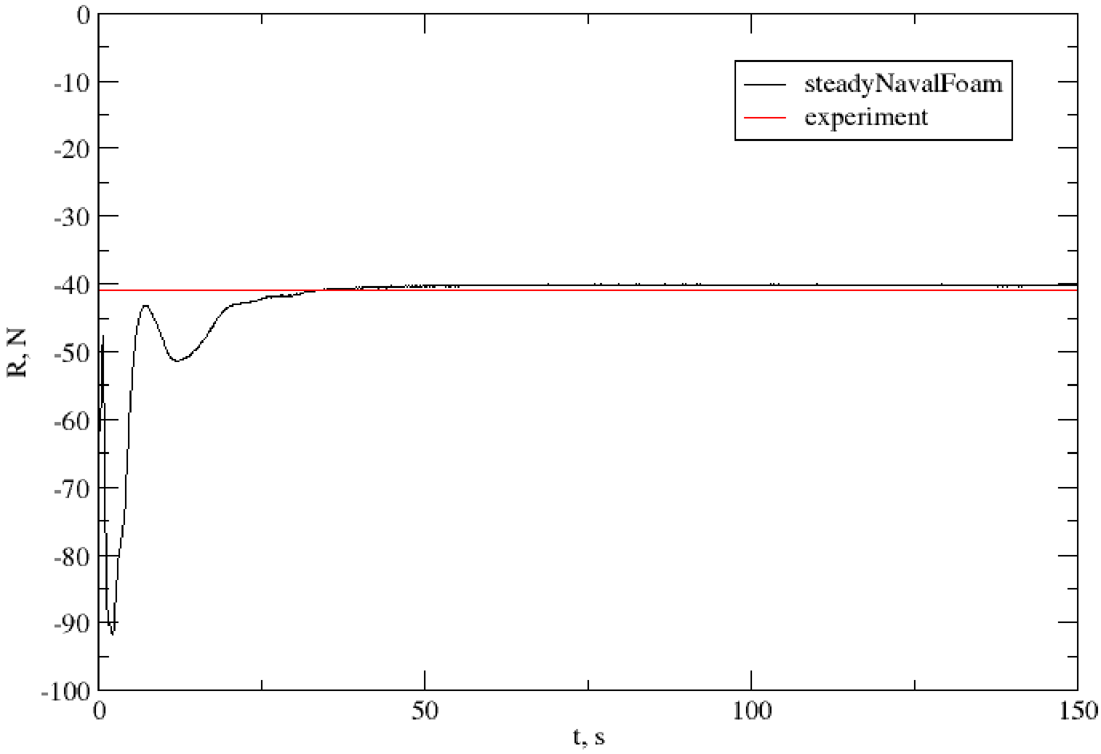

Fig 4. Drag force convergence and experimental comparison, KCS, $Fn = 0.26$

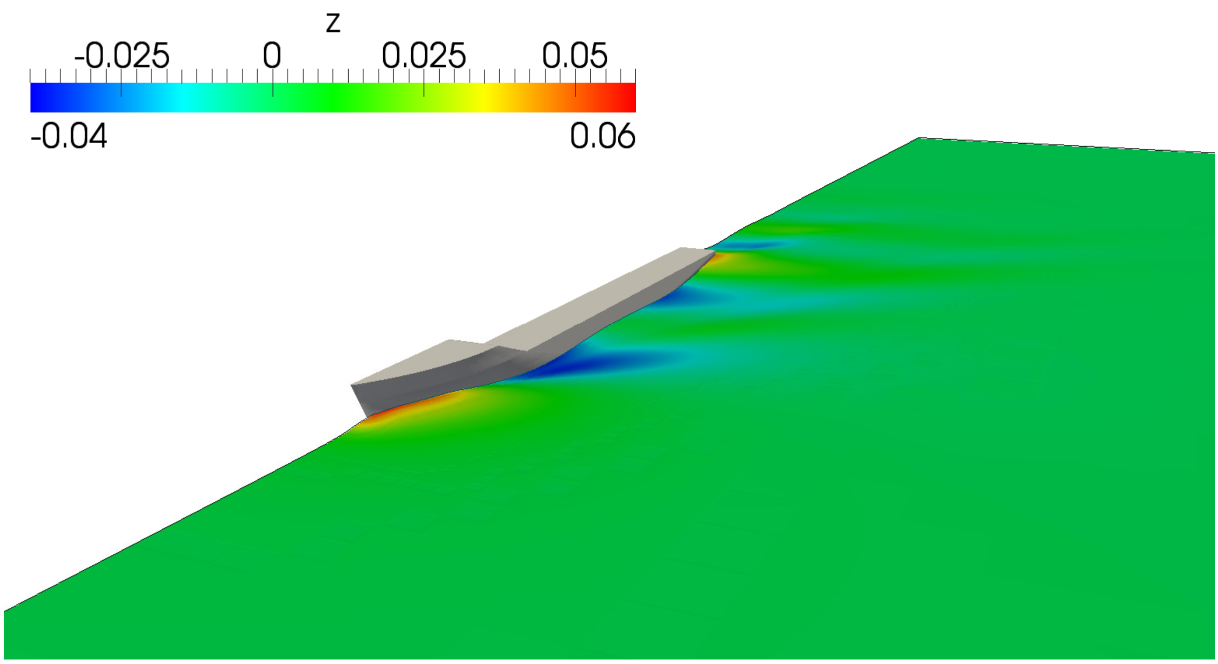

Fig 5. Steady state free surface for KCS, perspective view

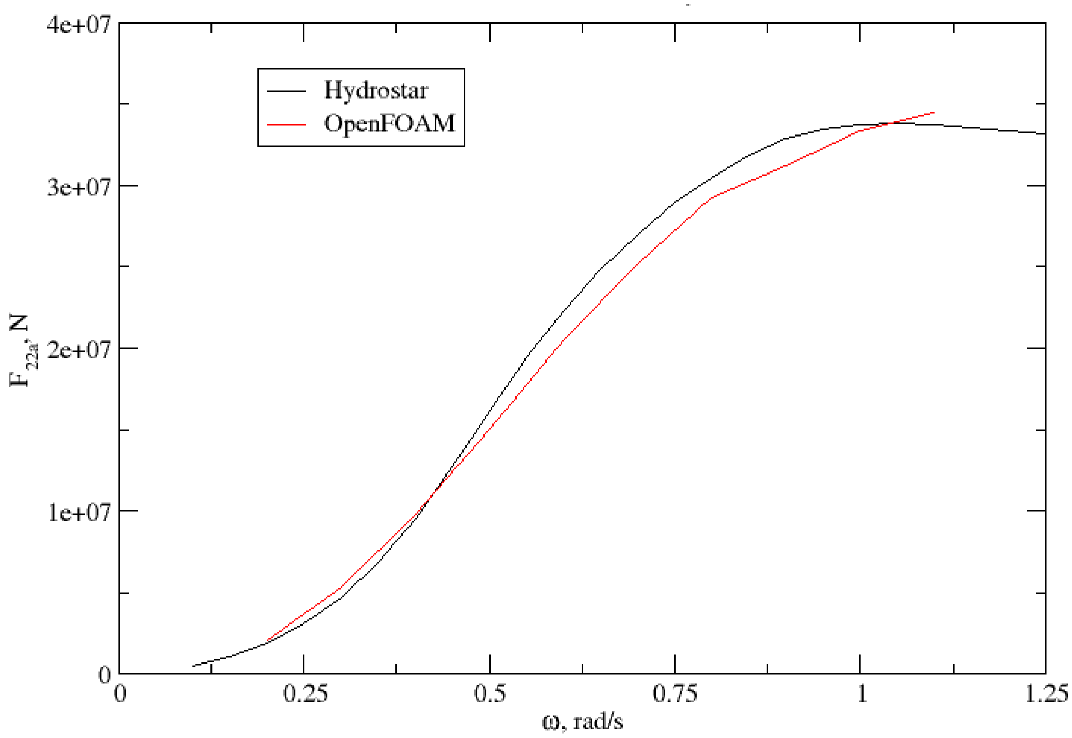

Fig 6. Sway force amplitude with respect to angular frequency

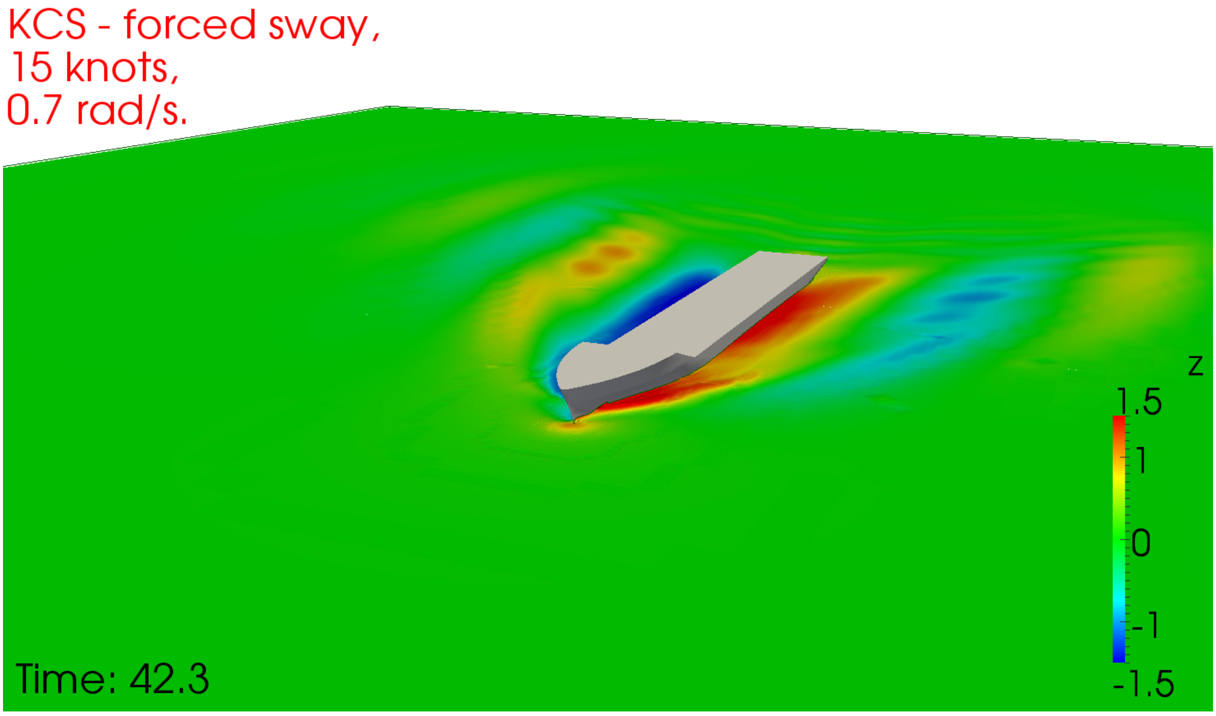

Fig 7. Free surface in random time instant for forced sway simulation

6. Seakeeping simulations of KCS in head waves

Finally, the KCS model as described in section 4 is used for seakeeping simulation. Since the model is symmetric, 2 Degrees of Freedom (DOF) were calculated (heave and pitch). Extension to 6 DOF simulations is straightforward; one only needs to provide the inertia tensor for the ship. Forward speed of 2.196 m/s is considered. Regular head waves are simulated using relaxation zones. Wave period is set to 3 s, and wave height to 0.25 m, compared to model length of 7.28 m.

Second order Stokes waves are used in this simulation. Fig 8. presents the variation of surge force during the simulation, whereas Fig 9. shows heave motion of the centre of gravity compared to the initial vertical centre of gravity (VCG). Finally, Fig 10. presents the free surface at a given instant in time. There are no experimental data available for comparison.

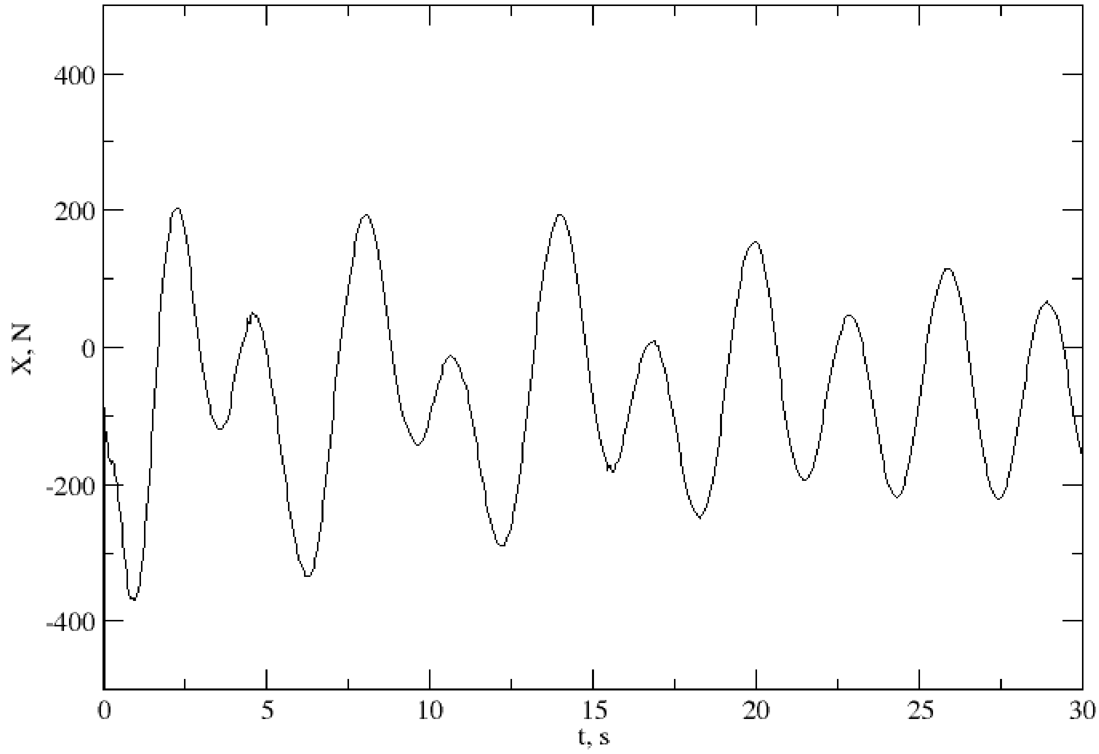

Fig 8. Surge force signal

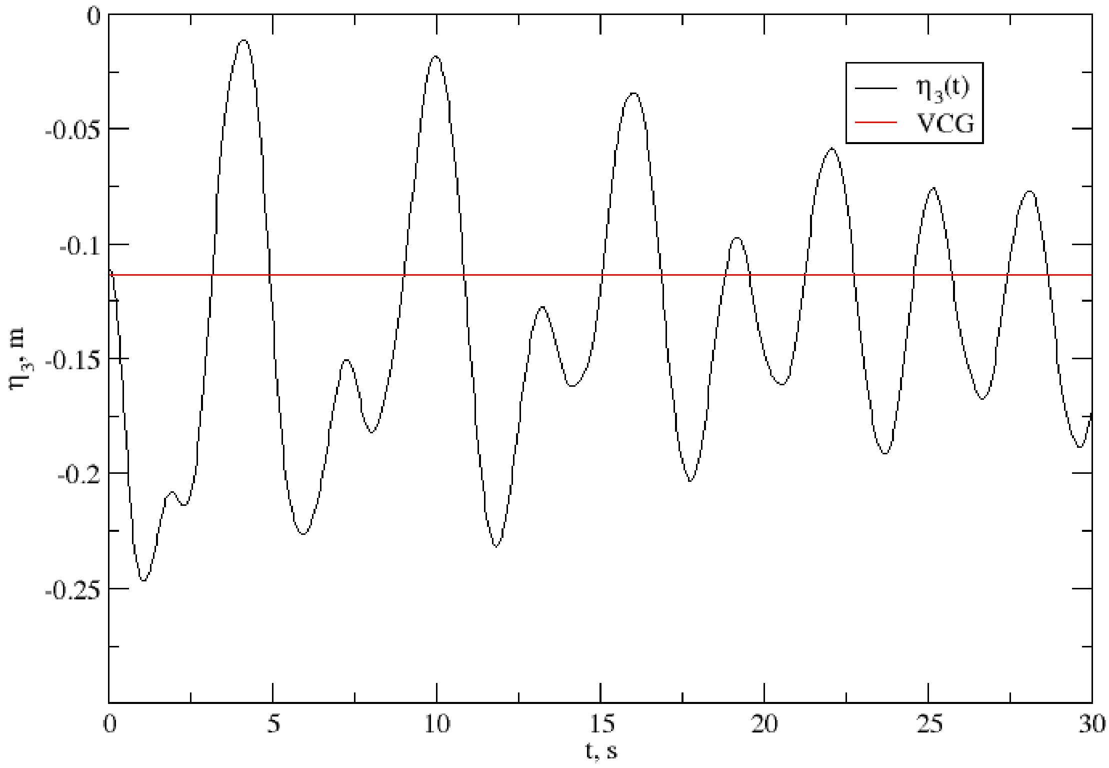

Fig 9. Heave motion signal

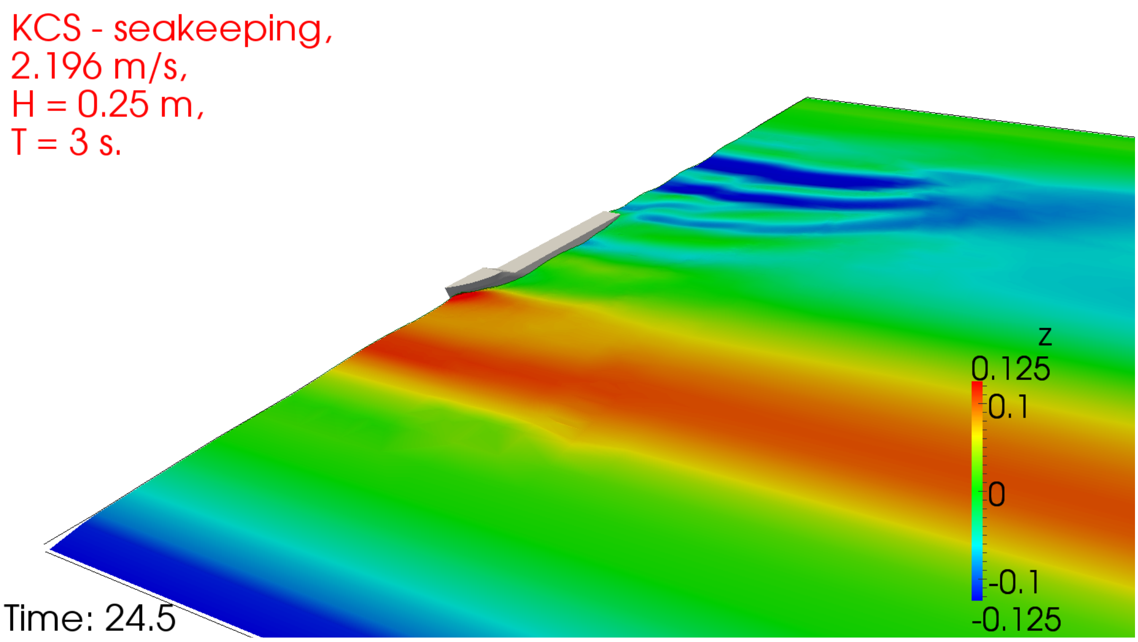

Fig 10. Free surface in random time instant for seakeeping simulation.

7. Conclusion

This paper briefly presents various capabilities of OpenFOAM in marine hydrodynamics. Results of steady state resistance are in good agreement with experimental results. Forced sway simulations of the KCS hull are also in good agreement when compared to BEM solution. It should be noted that simulations presented here include non-linear, viscous and turbulent effects which are neglected in BEM. Furthermore, various seakeeping simulations can be carried away with relative ease. These kinds of simulations can be used to obtain added resistance in waves, excessive motion amplitudes, green water effects, etc. It should also be noted that the improved numerics have lowered needed computational resources which can be seen from the reported CPU times.

References

[1] WELLER, H. G., TABOR, G., JASAK, H.: "A tensorial approach to computational continuum mechanics using object oriented techniques", Computers in Physics (1998)12, p. 620-631.

[2] UBBINK, O., ISSA, R. I.: "A method for capturing sharp fluid interfaces on arbitrary meshes", Journal of Computational Physics (1999)153, p. 26-50.

[3] RUSCHE, H.: "Computationl Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions", Ph.D. thesis, Imperial College of Science, Technology & Medicine, London, 2002.

[4] WILCOX, D. C.: "Turbulence Modeling for CFD", DCW Industries, Inc., La Canada, CA, 2006.

[5] MENTER, F. R., KUNTZ, M., LANGTRY, R.: "Ten Years of Industrial Experience with the SST Turbulence Model", Turbulence, Heat and Mass Transfer (2003)4, p. 625-632.

[6] JASAK, H.: "Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows", Ph.D. thesis, Imperial College of Science, Technology & Medicine, London, 1996.

[7] JACOSBSEN, N. G., FUHRMAN, D. R., FREDSØE, J.: "A Wave Generation Toolbox for The Open – Source CFD library: OpenFOAM", International Journal for Numerical Methods in Fluids (2012)9, p. 1073-1088.

[8] ...: "Gothenburg 2010, A Workshop on CFD in Ship Hydrodynamics", Available at http://www.insean.cnr.it/sites/default/files/gothenburg2010/index.html, 25 March 2014.

[9] ...: "SIMMAN 2014", Available at http://www.simman2014.dk/cms, Available at 25 March 2014.

[10] ...: "HYDROSTAR for Experts, v6.11 – User Manual", Bureau Veritas, Paris, 2010.