CFD Validation of a Container Ship in Calm Water and Head Seas

Inno Gatin[1], Vuko Vukčević[1] and Hrvoje Jasak[1]

[1] *University of Zagreb, Zagreb/Croatia, inno.gatin@stud.fsb.hr, {vuko.vukcevic, hrvoje.jasak}@fsb.hr ,*

1 Introduction

Added resistance of ships in waves is one of increasingly important problems in naval engineering due to energy efficiency regulations. In this work, validation of the Naval Hydro pack in OpenFOAM is performed by conducting simulations of a KRISO container ship (KCS) in calm water and head waves. Steady resistance and dynamic sinkage and trim at different Froude numbers are compared to experimental data. Mesh refinement study has been carried out for design Froude number in order to asses numerical uncertainty. Seakeeping simulations of the ship in head waves are carried out for a number of different wave parameters at design Froude number. Added resistance is compared with experimental results. All simulations are performed with fully non–linear, turbulent, two–phase CFD solver. Wave modelling is performed using Spectral Wave Explicit Navier–Stokes Equations (SWENSE) [1] with implicit relaxation zones [5] that are used to prevent wave reflection. In addition to the validation runs, 3-D freak wave simulation encountering a KCS has been performed using the SWENSE solver coupled with a directional Higher Order Spectrum (HOS) [2] method for nonlinear wave propagation.

2 Mathematical model

In this section mathematical model of incompressible two-phase flow is presented. Two–phase model is based on a single set of mixture equations taking into account jump conditions at the free surface implicitly [4]. Volume of Fluid (VOF) method will be used for interface capturing [9] in calm water simulations, while the Level Set (LS) method will be used for head waves simulations, where SWENSE procedure is employed.
Three sets of governing equations are used to model the two phase flow: continuity equation, momentum equation and interface capturing equation. The continuity equation, used to formulate dynamic pressure equation, states:

$$\nabla \bullet \mathbf{u} = 0. \tag{1}$$

Mixture momentum equation reads:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \bullet (\mathbf{u}\mathbf{u}) - \nabla \bullet (\nu_e \nabla \mathbf{u}) = -\frac{1}{\rho}\nabla p_d\,, \tag{2}$$

where ν_e stands for effective dynamic viscosity, allowing for the general turbulence models [11]. Interface capturing equations used in the calculations can be found in Rusche (2002) [7] for VOF method, and in Sun & Beckermann (2007) [8] for LS method.
Apart from the governing equations, special attention is given to the jump conditions at the free surface. In the present model, normal stress balance is modelled exactly, while the tangential stress is taken into account approximately. Surface tension is neglected. This approach is justified for large scale two–phase problems encountered in naval hydromechanics. Normal stress balance yields two free surface jump conditions [4]. First condition prescribes the dynamic pressure jump across the free surface:

$$p_d^- - p_d^+ = -(\rho^- + \rho^+)\mathbf{g}\bullet\mathbf{x}\,, \tag{3}$$

where superscript "+" denotes the heavier fluid (water), while superscript "−" denotes the lighter fluid (air). Second condition prescribes the jump of dynamic pressure gradient over density to be zero:

$$\frac{1}{\rho^-}\nabla p_d^- - \frac{1}{\rho^+}\nabla p_d^+ = 0. \tag{4}$$

Jump conditions are used to discretize dynamic pressure terms (pressure gradient and laplacian) at the free surface.

3 Numerical model

Numerical model is based on second–order accurate polyhedral FV method used in foam–extend. Rigid body motion is solved using quaternion based six degrees of freedom (6DOF) equations. Mesh is modelled as a rigid body with special boundary conditions. Coupling of pressure, velocity, free surface and 6DOF equations is performed in a segregated manner using PIMPLE algorithm.

4 Numerical simulations

In this section simulation results regarding KCS model are compared to experimental data. Steady state resistance with dynamic sinkage and trim is simulated for a range of Froude numbers. Seakeeping simulations are performed in regular head waves with different incident wave parameters at design Froude number. Finally, a simulation of a freak wave encountering a full scale KSC is shown.

4.1 Steady resistance, simulations with dynamic sinkage and trim

CFD results computed on 4.6 million cells unstructured mesh are compared with experimental data for following Froude numbers: 0.1516, 0.1949, 0.2274, 0.2599 and 0.2816. KCS model particulars and experimental settings details are available at Tokyo Workshop On CFD in Ship Hydrodynamics website [6]. Symmetry boundary condition is used at the central vertical longitudinal plane, and only half of the domain is simulated. Average mesh non–orthogonality is 6.6 degrees while the maximum is 76.1 degrees. Results comparison for total resistance coefficient is shown in Figure 1. Dynamic sinkage and trim comparison is shown in Figure 2 and Figure 3, respectively.

Grid convergence study is performed for 0.282 Froude number. Table 1 shows the total resistance coefficient, sinkage and trim obtained with different grid sizes. Relative error compared to experimental result is also presented. Relative error is calculated as $\epsilon = (C_{tEFD} - C_{tCFD})/C_{tEFD}$. Calculated order of spatial accuracy for total resistance [10] is $p = 3.98$ for three finest grids, while grid uncertainty is 0.029×10^{-3}.

4.2 Seakeeping simulations in regular head waves

Added resistance is compared with experimental data for various incident wave parameters. Simulations are performed for design Froude number and head waves. Comparison is performed in Fourier space, comparing individual harmonic amplitudes to experimental data [6]. KCS model particulars can be found on Tokyo Workshop On CFD in Ship Hydrodynamics website [6]. Figure 4 shows comparison of mean, first and second order of total resistance coefficient for different incident wave parameters (λ/L_{pp}).

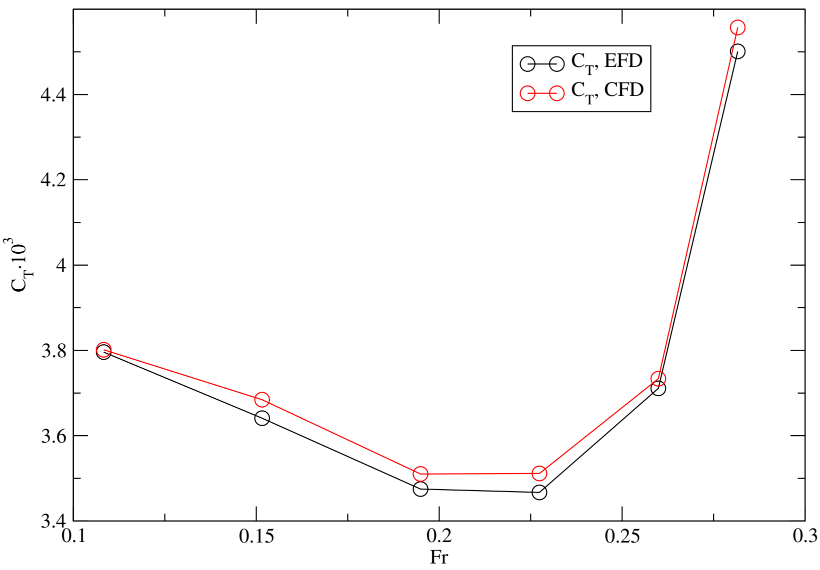


Figure 1: Comparison of total resistance coefficient for a range of Froude numbers.

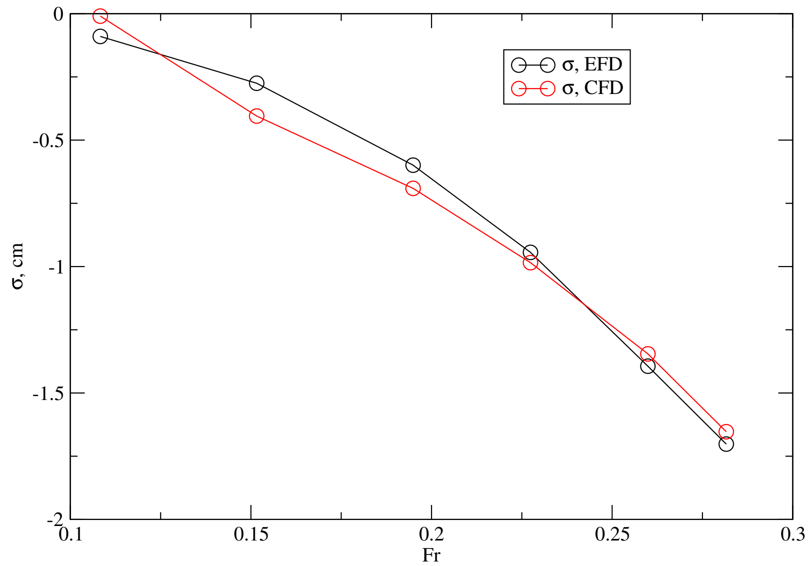


Figure 2: Comparison of dynamic sinkage for a range of Froude numbers.

Table 1: Grid convergence study results.

Grid No.	1	2	3	4	EFD
$C_t \times 10^3$	4.837	4.736	4.608	4.560	4.501
$\epsilon_{C_t}, \%$	-7.476	-5.234	-2.381	-1.313	/
σ, cm	-1.665	-1.650	-1.655	-1.653	-1.702
$\epsilon_\sigma, \%$	2.194	3.067	2.756	2.867	/
$\tau, ^o$	-0.145	-0.147	-0.147	-0.146	-0.159
$\epsilon_\tau, \%$	8.813	7.264	7.847	7.974	/

It can be seen in Figure 4 that the trend of all the curves is well captured. Mean values obtained with CFD generally overshoot the experimental data, which is consistent with the

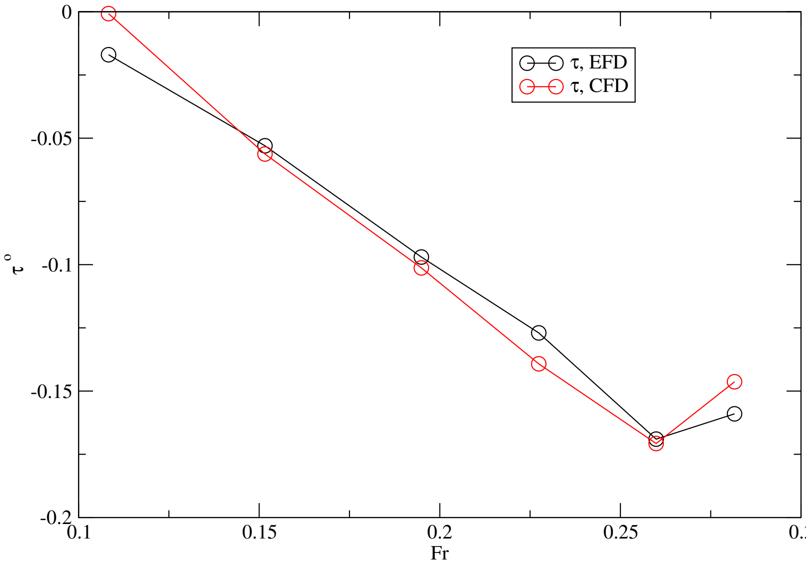


Figure 3: Comparison of dynamic trim for a range of Froude numbers.

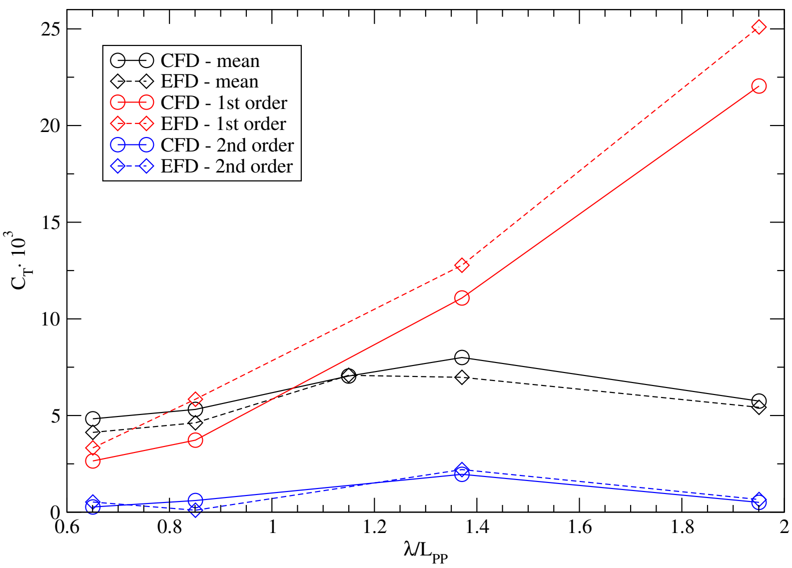


Figure 4: Comparison of total resistance coefficient for various incident wave parameters.

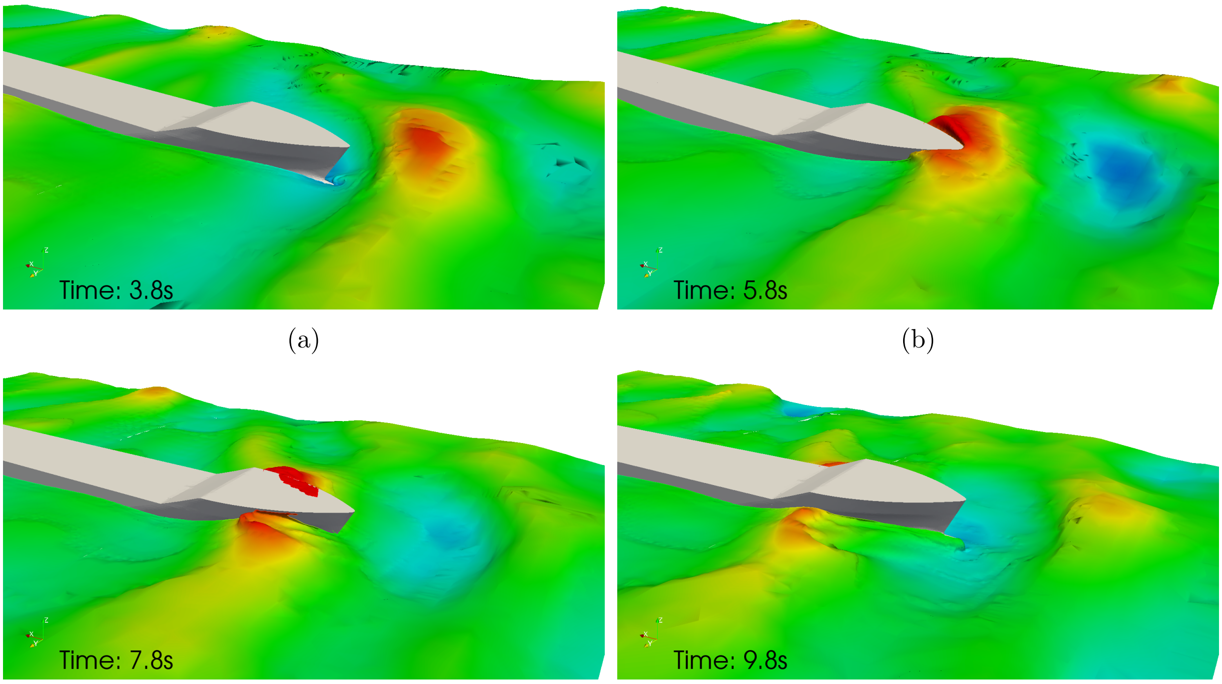
steady state resistance simulations. First order resistance is undershot for all cases.

4.3 Freak wave impact on a full scale floating KCS

A three-dimensional, realistic freak wave is initialized using long time evolution of wave spectrum with Higher Order Spectrum (HOS) potential nonlinear wave theory [2] implemented in Naval Hydro pack. Initial condition used for HOS simulation is JONSWAP directional spectrum. Wave spectrum parameters are: $H_s = 10.5$ m, $T_p = 9.5$ s, and directionality constant $n = 8$ [3]. Freak wave height is $H = 21.8$ m. Simulation is performed for full size KCS, Table 2. Figure 5 sequentially shows the encounter of the freak wave on the KCS.

Table 2: Full scale KCS characteristics.

L, m	B, m	D, m	T, m	x_G, m	y_G, m	KG, m	r_x, m	r_y, m	r_z, m
230	32.2	19	10.8	111.6	0	14.32	12.9	57.5	57.5



(a) (b) (c) (d)

Figure 5: Three–dimensional freak wave encountering a full scale KCS.

5 Conclusion

Simulations of KCS in calm water and head waves are carried out and the results are compared with experimental data. Steady state simulations show good agreement for resistance and dynamic sinkage and trim. Larger discrepancies of dynamic sinkage and trim for lower Froude numbers are caused by coarse mesh considering the absolute values of sinkage and trim. Preliminary results of added resistance in head waves are compared to experimental data, being an important design criteria. Comparison showed that added resistance can be predicted fairly accurately. Finally, an example of three–dimensional freak wave simulation is performed to show the capability of the HOS wave theory coupled with CFD simulation using SWENSE method.

References

[1] Ferrant, P. and Gentaz, L. and Le Touzé, D. A new ranse/potential approach for water wave diffraction. *Proc. Numerical Towing Tank Symposium*, 2002.

[2] I. Gatin. Master Thesis; Implementation and validation of the Higher Order Spectrum (HOS) method for Simulation of Surface Waves. *Faculty of Mechanical Engineering and Naval Architecture, Zagreb*, 2015.

[3] L. H. Holthuijsen. *Waves in Oceanic and Costal Waters.* Delft University of Techonlogy and UNESCO-IHE, 2007.

[4] J. Huang, P. M. Carrica, and F. Stern. Coupled ghost fluid/two–phase level set method for curvilinear body–fitted grids. *Int. J. Numer. Meth. Fluids*, 44:867–897, 2007.

[5] H. Jasak, V. Vukčević, and I. Gatin. Numerical Simulation of Wave Loads on Static Offshore Structures. In *CFD for Wind and Tidal Offshore Turbines*, pages 95–105. Springer Tracts in Mechanical Engineering, 2015.

[6] National Maritime Reasearch Inistitute (NMRI). Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics. `http://www.t2015.nmri.go.jp/`, 2015. [Online; accessed 20 August 2015].

[7] H. Rusche. *Computational Fluid Dynamics of Dispersed Two - Phase Flows at High Phase Fractions*. PhD thesis, Imperial College of Science, Technology & Medicine, London, 2002.

[8] Y. Sun and C. Beckermann. Sharp interface tracking using the phase–field equation. *Journal of Computational Physics*, 220:626–653, 2007.

[9] O. Ubbink and R. I. Issa. A method for capturing sharp fluid interfaces on arbitrary meshes. *Journal of Computational Physics*, 153:26–50, 1999.

[10] H. K. Versteeg and W. Malalasekera. *An introduction to Computational Fluid Dynamics: The finite volume method*. Pearson Education Limited, 1995.

[11] D. C. Wilcox. *Turbulence Modeling for CFD*. DCW Industries, 1993.