URANSE simulation for the Seakeeping of the KVLCC2 Ship Model in Short and Long Regular Head Waves

A Bekhit and A Lungu
“Dunarea de Jos” University of Galati, Faculty of Naval Architecture, 47 Domneasca Street, 800008, Galati, Romania

E-mail: adham.bekhit@ugal.ro

Abstract. In the present study, the vertical motions and the added resistance in waves of the KVLCC2 ship model are predicted numerically in regular head wave, for a single wave height and different wave lengths, including long and short waves. The numerical simulation is performed by making use of the ISIS-CFD solver of the commercial software Fine™/Marine provided by NUMECA, where the discretization in space is based on finite volume method using unstructured grid. The unsteady RANS are numerically solved, while the turbulence is modelled by making use of the $k$-$\omega$ SST model. The free-surface is captured through an air-water interface based on the Volume of Fluid method. For validation purposes, the computed solutions are compared with the available tank test data existing in the public domain. A systematic grid convergence study based on Richardson Extrapolation method is performed for a single wave case on three different grid resolutions, as an attempt of predicting the uncertainties in the numerical solution. A thorough investigation for the free-surface topology at different encountering moments is also presented to prove not only the accuracy of the solver, but also its robustness.

1. Introduction
Ship performance in a seaway is crucially important to be accurately predicted. Significant ship motions can influence the safety of the ship, while ship motion in general should be taken into consideration to provide a comfortable level for the crew operations on board. On the other hand, added resistance in waves can have a recognized impact on the propulsion efficiency and a significant consequence on fuel cost. In addition, the International Maritime Organization (IMO) recently issued new regulations to control the fuel emission under the Marine Environment Protection Committee (MEPC), which requires an optimization of ship performance from powering point of view in accordance with the Energy Efficiency Design Index (EEDI) and Energy Efficiency Operational Indicator (EEOI). This compels a naval architect to provide an optimum and rather accurate prediction for the added resistance in waves, to achieve a suitable balance between the required ship powering and fuel cost and emissions. In general, ship performance in waves can be predicted either experimentally or numerically. The numerical prediction of ship performance in waves can be done either using potential flow theory or Computational Fluid Dynamics (CFD, hereafter). There are two major analytical approaches in potential flow methods which are used to calculate the added resistance: the far-field method and the near-field method. The far-field method is based on the added resistance computed from the wave energy and the momentum flux generated by a ship and is
evaluated across a vertical control surface of infinite radius surrounding the ship. The near-field method estimates the added resistance by integrating the hydrodynamic pressure on the body surface [1]. Despite the popularity of these approaches and their easy implementation, they have some drawbacks due to ignoring the viscous effect, which becomes more significant for high Froude numbers and for viscous influenced motions, such as roll motion, where the damping effect is significantly a viscous-based phenomenon.

Most recently, as the computational capacity has become more powerful and accessible, CFD approaches started to become more popular for predicting added resistance and motions in waves. CFD method has an advantage over the potential method because extreme wave condition such as green water, slamming impact loading, deck load, whipping, breaking waves, dynamic structural loading, etc. cannot be handled properly with potential flow assumption. In such situation, RANS simulation can provide a more reliable prediction [2]. Nonetheless, CFD solvers require very significant computational time compared to the potential based solvers.

Many researches were presented based on CFD in the scope of solving seakeeping problems, including incident waves and prescribed or free pitch, heave and roll motions. The very first attempt to solve ship motion in waves was presented in [3], which suffered some problems regarding accuracy of the free-surface due to the limited grid quality. In Tokyo Workshop 2005, a forward speed diffraction case was proposed for the DTMB ship model in head waves. Four different results were presented in the Workshop with a good agreement in predicting resistance and heave forces, while the error for pitching moment was quite significant [2]. In the past decade, a well-recognized development in the accuracy was achieved by CFD for seakeeping prediction including ship motion and added resistance in waves. The seakeeping performance of the DTMB ship model was predicted and presented in [4] including ship motion and resistance in waves. The accuracy of the obtained results was validated against the tank test results and showed a very good agreement, while the consistency of the numerical scheme was assessed through a systematic verification study. In Gothenburg Workshop (G2010) the seakeeping performances of the KVLCC2 ship was presented by five groups of results for the heave and pitch motion prediction based on different CFD commercial codes, while only two participations were presented for three degrees of freedom motion in waves including surge, heave and pitch [5]. The obtained results showed a good agreement with experimental results, especially for ship motion, while for resistance forces, the error was slightly larger. The added resistance and vertical motions of the KVLCC2 ship model at different Froude number was presented in [6], the scope was to investigate the maximum response, resonance conditions and wave excitation forces; besides, a closer insight on the local velocity flow field was also presented and validated against experimental data showing a reasonable agreement. Numerical investigation of added resistance and vertical motion of the KVLCC2 ship at various ship speeds and wave steepness was performed based on both URANS and 3-D potential method and presented in [7]. The results were validated at design speed at various wave conditions to show the accuracy of the numerical approach. A further investigation at different speeds and wave conditions were also performed using CFD method, with an attentive interest on the relationship between non-linearity of the added resistance and ship motions with the varying wave steepness. The CFD tools were recently used to enhance ship performance in waves by geometrical modifications as the researches presented in [8-10].

In this study, a comprehensive investigation for seakeeping performance of the KVLCC2 ship model in regular head waves is numerically assessed for a single wave height and three different wave lengths, including long and short waves compared to the ship length. The scope is to predict the vertical motions and the added resistance in waves. The study also includes a systematic verification study based on three computational grids to assess the accuracy of the numerical solutions and to predict the associated uncertainties. Validation of the obtained numerical results is done by direct comparison with the available experimental data that were presented in the G2010 Workshop. Finally, a special focus on the free-surface topology for different wave conditions is also presented at different wave encountering periods.
2. Ship geometry and computational conditions

The case study for this paper is applied on the modified hull of the Korea Research Institute of Ships and Ocean Engineering (KRISO) Very Large Crude Carrier (KVLCC2, hereafter). The ship was conceived to provide benchmark data for both explication of flow physics and CFD validation for a modern tanker ship with bulb bow and stern. The full scale ship does not exist; however, several models were built and tested by various internationally well recognized towing tank organizations such as National Maritime Research Institute (NMRI) and Osaka University (OU) in Japan, the Istituto Nazionale per Studi ed Esperienze di Architettura Navale (INSEAN) in Italy, Norwegian University of Science and Technology (NTNU) and Hyundai Maritime Research Institute (HMRI) in Korea. Several experiments were performed, including ship resistance, free-surface, local flow configurations, seakeeping and maneuverability.

In this case study, the INSEAN ship model of scale 1/100 is used for validating this simulation, which has a length between perpendiculars of \( L_{pp} = 3.2 \) m and a draft of 0.208 m. Full details of the ship characteristics are tabulated in table 1, while the geometry of the ship is depicted in figure 1. This model has the same scale as the one used in OU, except that the depth is \( D = 0.35 \) instead of 0.3, where the latter encountered some green water on the deck after some pilot computations that were performed by the authors; the same phenomena was also observed in the tank test in OU for longer waves cases, as it is stated in [2]. Since this study does not investigate the non-linear phenomena, the higher depth was chosen to avoid deck greening.

Table 1. Particulars of the KVLCC2 ship model.

| Parameter                  | Unit | Value |
|----------------------------|------|-------|
| Length, \( L_{pp} \)       | m    | 3.2   |
| Beam, \( B \)              | m    | 0.58  |
| Draft, \( T \)             | m    | 0.208 |
| Depth, \( D \)             | m    | 0.35  |
| Block coefficient, \( C_b \) | -    | 0.506 |
| Mid-ship section coefficient, \( C_M \) | -    | 0.998 |
| Displacement, \( \varpi \) | m\(^3\) | 0.3126 |
| Moment of inertia, \( K_{o/B} \) | -    | 0.4   |
| Moment of inertia, \( K_{o y/L_{pp}} \) \( K_{o z/L_{pp}} \) | m \( L_{pp} \) | 0.25 |

Figure 1. Hull of the KVLCC2 model.

Three computation conditions are performed in this study, including short and long waves with respect to the length of ship \( L_{pp} \). A single wave height is applied for the three cases \( H_w = 0.06 \) m. These three conditions complies with the case “1.4a” seakeeping in regular head waves that was presented in G 2010 Workshop, which is corresponding to the ship sailing in head waves at the design velocity \( F_r = 0.142 \) including two degrees of freedom represented in heave and pitch (FRz0) [5]. Full details of the computation conditions are summarized in table 2.

Table 2. Computational cases and wave characteristics.

| Computational case | \( \lambda_w/L_{pp} \) | \( H_w \) | \( T_w \) | \( f_w \) | \( T_c \) | \( f_c \) |
|--------------------|------------------------|----------|----------|----------|----------|----------|
| Case 1             | 0.6                    | 0.06     | 1.1089   | 0.9018   | 0.7598   | 1.3161   |
| Case 2             | 1.1                    | 0.06     | 1.5015   | 0.6660   | 1.1211   | 0.9820   |
| Case 3             | 1.6                    | 0.06     | 1.8109   | 0.5522   | 1.4132   | 0.7076   |
3. Numerical approach
All the computations are performed by making use of the ISIS-CFD solver of the commercial software Fine\textsuperscript{TM}/Marine, provided under NUMECA’s umbrella. The solver is based on the finite volume method to build the spatial discretization of the transport equations to solve the incompressible unsteady RANS equations. Closure to turbulence is achieved by using the $k$-$\omega$ SST model with a near wall resolution based on wall function, setting the first point close to the wall at $y^+ = 32$. A three-dimensional face-based method is generalized to build unstructured meshes. Velocity-pressure coupling is achieved using a SIMPLE like approach, where the velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation. Free-surface flow is simulated with a multi-phase flow approach based on the Volume of Fluid (VOF) free-surface capturing method. Incompressible and non-miscible flow phases are modeled through the use of conservation equations for each volume fraction of phase/fluid [11]. Convection and diffusion terms in the RANS equations are discretized by a second-order upwind scheme and a central difference scheme, respectively. Implicit scheme is applied for time discretization. Second order three-level time scheme is employed for time-accurate unsteady computation [2].

3.1. Governing equations
For incompressible flows including external forces, the averaged continuity and momentum equations 1 and 2, respectively, can be written in tensor form, in the Cartesian coordinate system as

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

$$\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_i \bar{u}_j + \mu \overline{u}_i u_j' \right) = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j}$$

where $\bar{u}_i$ is the relative averaged velocity vector of flow between the fluid and the control volume, $u_i' u_j'$ is the Reynolds stresses, $\bar{p}$ is the mean pressure and $\bar{\tau}_{ij}$ is the mean viscous stress tensor components for Newtonian fluid under the incompressible flow assumption, and it can be expressed as in equation 3

$$\bar{\tau}_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)$$

3.2. Computational domain
The computational domain is having a rectangular prism configuration as depicted in figure 2. Dimensions of the computational domain in the $(x-y-z)$ direction are $(7.0L_{ref}, -2.0L_{ref}, 6.0L_{ref})$, where $L_{ref} = \text{max}(L_{pp}, \lambda)$, distributed as $2.0L_{ref}$ upstream, $4.0L_{ref}$ downstream, $2.0L_{pp}$ on the side, $4.0L_{ref}$ underneath and $2.0L_{ref}$ above the undisturbed free-surface level, which is located at $z=0.208 \text{m}$ from baseline, as it is highlighted in blue in figure 2. Taking into account the symmetry of the hull and considering the fact that only vertical motion is taken into consideration with only head wave condition, only half of the ship is presented in the numerical simulations to simplify the problem and reduce the number of grid cells required, and hence the computational effort.

The boundary conditions are shown in figure 2 for the computational domain boundaries, which include a velocity inlet upstream with a wave generator to create the corresponding wave, while the far-field is applied on the downstream. The mirror boundary condition is applied on both sides of the domain at $y=0$ and $y=2.0L_{pp}$; the first is for the symmetry condition, as previously described, while the latter is applied to eliminate any numerical reflections for the wave at the side boundary. The prescribed pressure condition is applied on the top and bottom boundaries.
3.3. **Computational grid**

Computational grids are generated by using the unstructured hexahedral grid generator HEXPRESS™ which is included in the Fine™/Marine package. An individual grid is generated for each wave condition to maintain the consistency of the numerical approach, while the balance between the different simulation conditions is achieved by providing similar grid generation and refinement parameters. The grid convergence study is applied on the computational case 2, where the wave length $\lambda_w = 1.1 L_{PP}$. For this purpose, three geometrically similar grids are generated with systematic gradually increased refinement ratios to insure the grid similarity. The coarse grid was first generated, then a refinement criteria was imposed to generate similar grids with an average refinement ratio $r_G = 1.71$. The computational grids details are tabulated in table 3, where M1, M2 and M3 are indicating the fine, medium and coarse grid resolutions, respectively.

| Computational case | Case 1 | Case 2 | Case 3 |
|--------------------|-------|-------|-------|
| Number of grid cells (M) | 10.646 | 14.562 | 8.531 | 4.985 | 3.799 |

The free-surface is refined on the entire domain using two different refinement boxes. The first starts at the upstream boundary and is extended for one wave length behind the ship. The cell size is chosen for this refinement zone in $x$- and $y$- directions such that $\Delta x = \lambda_w / 50$, $\Delta y = 2.0 \Delta x$, while $\Delta z = H_w / 16$. The refinement depth in $z$-direction is extended for 3.0 $H_w$ equally distributed above and beneath the undisturbed free-surface. The second refinement box, which is extended downstream for 3.0$\lambda_w$ is used as a numerical damping zone, where the cell size is coarsened gradually in $x$- and $y$- directions only, by a factor of 4.0, then 8.0 nearby the exit boundary, while the $z$- direction refinement is maintained unchanged. In order to increase the accuracy of predicting the near-ship waves, an extra conical shape Kelvin-pattern wave refinement segment is added nearby the hull with a 40° opening angle to provide sufficiently fine grid for the hull-wave interaction prediction. The cell size in this zone is chosen such that $\Delta x = \Delta y = \lambda_w / 100$ and $\Delta z = H_w / 20$. The grid configuration is depicted in figure 3 for the fine grid in Case 2, showing the forward, aft and free-surface grids.

![Figure 2. Computational domain, dimensions and boundary conditions.](image-url)
3.4. Simulation technique

All the computations are performed on a High Performance Computing (HPC) machine with 120 cores at 3.3 GHz. The simulation is performed for 30 seconds or until the ship encounters 15 waves. All cases are performed with 12 non-linear iterations and the time step $\Delta t$ is chosen to be 250 time steps per wave period, with fourth order convergence criteria. The flow is accelerated for one second to avoid numerical overshoots in the beginning of the simulation. The wave characteristics are applied on the inlet boundary, including the corresponding wave length, height and period. A pilot computation was performed in the beginning to insure the solution stability, convergence and to check the accuracy of wave generator before performing all the computations. The solution becomes periodic after about 7-10 seconds in all the computational cases. The total simulation time for each case ranged between 32–82 physical hours, which are definitely dependent to the grid resolution.

![Simulation technique](image)

**Figure 3.** Discretization grid: (a) at the ship extremities & (b) Free-surface refinement.

4. Results and discussion

4.1. Case 1

Case 1 represents the short wave length condition as described before, where $\lambda_w=0.6L_{PP}$, with the shortest wave period and the highest frequency. Simulation is performed for about 17 seconds. The Experimental data (EFD) were not introduced for this case in G2010 Workshop. For that reason, the results here are compared with the average value from all the five participants in the Workshop. Table 4 includes the results for the 0th and 1st amplitudes for ship resistance, heave and pitch motions, while the time history of the ship resistance is plotted in figure 4. It is worth mentioning that there is no phase analysis performed in this simulation, only amplitudes of forces and motions are taken into consideration.

The results obtained in this study seem to be within a very close range compared to the results obtained in G2010 Workshop, especially for the resistance case, where the percentage relative difference between the average results obtained in G2010 and the current study for ship resistance is 0.01% and 8.6% for 0th and 1st amplitudes, respectively. Though the difference for the 1st amplitude apparently seems to be significant, one can observe that the value is still within the same range obtained by UIOWA. The same trend can be observed for the heave and pitch values. Worth mentioning that the negative values for heave and pitch means that: a positive heave value is defined upwards and a positive pitch value is defined bow up.
Table 4. Case 1 results compared to the obtained results in G2010 Workshop [5].

|               | $C_T \times 10^3$ | $z$ [mm] | $\theta$ [°] |
|---------------|-------------------|----------|---------------|
|               | 0\textsuperscript{th} Amp. | 1\textsuperscript{st} Amp. | 0\textsuperscript{th} Amp. | 1\textsuperscript{st} Amp. | 0\textsuperscript{th} Amp. | 1\textsuperscript{st} Amp. |
| EFD           | -                 | -        | -             | -             | -             | -             |
| ECN (ISISCFD) | 7.758             | 43.410   | -3.360        | 1.850         | -0.120        | 0.144         |
| ECN (ICARE)   | 7.550             | 45.200   | -3.350        | 2.080         | -0.117        | 0.152         |
| GL&UDE (Comet)| -                 | -        | -3.521        | 1.281         | -0.242        | 0.091         |
| GL&UDE (OpenFOAM) | -                 | -        | -2.386        | 1.633         | -0.109        | 0.109         |
| UIOWA (CFDShipIowa) | 7.796             | 40.204   | -3.324        | 1.584         | -0.116        | 0.118         |
| Average       | 7.701             | 42.938   | -3.188        | 1.686         | -0.1408       | 0.123         |
| Current study | 7.700             | 39.256   | -2.14         | 1.865         | -0.1084       | 0.106         |

Figure 4. Time history for computed resistance in waves in Case 1.

The free-surface is plotted for the four quarters of the encounter period in figure 5, where $t/T_e=0$ indicates the moment when the wave crest coincides with the forward perpendicular (F.P), $t/T_e=0.25$ indicates the mid-distance between the crest and trough, $t/T_e=0.5$ when the trough is at the F.P. and finally, $t/T_e=0.75$ indicates the mid-distance between the trough and crest. The free-surface topology shows that the radiated waves are generated at the fore shoulders and dissipated downstream causing a disturbance in the original wave system. The aft shoulders generate other wave systems that have limited interference with the original waves in the stern.

Figure 5. Free-surface topology at different encounter instants for Case 1.
4.2. Case 2
Case 2 represents the medium wave length condition as described before, where $\lambda_w = 1.1L_p$ with a medium wave period and medium frequency. Simulation is performed for about 23 seconds. The Experimental data (EFD) were presented for this case in G2010 Workshop. Validation of CFD results is done by direct comparison with the EFD results. Table 5 includes the results for the 0th and 1st amplitudes for ship resistance, heave and pitch motions, while the time history of the ship resistance in waves, heave and pitch are plotted in figure 6.

|                  | 0th Amp. | 1st Amp. | 0th Amp. | 1st Amp. | 0th Amp. | 1st Amp. |
|------------------|----------|----------|----------|----------|----------|----------|
| **C_T x 10^3**   |          |          |          |          |          |          |
| EFD              | 12.480   | 43.17    | -3.567   | 19.833   | -0.183   | 1.710    |
| M1               | 11.67    | 32.57    | -2.774   | 21.098   | -0.128   | 1.567    |
| $\varepsilon\%D$ | -6.49%   | -24.55%  | -22.23%  | +6.38%   | -30.06   | -8.36%   |
| M2               | 11.628   | 32.858   | -2.829   | 21.369   | -0.130   | 1.575    |
| $\varepsilon\%D$ | -6.82%   | -23.89%  | -20.689% | +7.74%   | -28.96%  | -7.89%   |
| M3               | 11.581   | 33.145   | -2.913   | 21.98    | -0.134   | 1.591    |
| $\varepsilon\%D$ | -7.20%   | -23.22%  | -18.33%  | +10.82%  | -26.77%  | -6.96%   |

**Table 5. CFD results for Case 2 compared to the EFD results [5].**

![Figure 6. Time history for computed resistance (top), heave (middle), pitch (bottom) for Case 2.](image-url)
The obtained results show an encouraging agreement with the available EFD data for the ship resistance 0\textsuperscript{th} amplitude. This value is used to compute the added resistance in waves, as it will be described later. The error for the 1\textsuperscript{st} amplitude of the resistance is apparently significant; yet, the error range is still in the same range of the results obtained in the G2010 Workshop \[5\]. The error for motions is significant for the 0\textsuperscript{th} amplitudes, while it seems to be within a reasonable range for the 1\textsuperscript{st} amplitudes. The aforementioned observation is still applied here, since the results are within the same range of the Workshop.

The grid convergence study based on Richardson Extrapolation is performed and presented in table 6 according to the rules and procedures provided in the ITTC recommended practice \[12\], where $R_G$ represents the grid convergence index, $S_1$ represents the CFD value for the finest grid, $D$ is the EFD value and $\delta_G$ stands for the grid error. The error $\varepsilon$ is computed based on the equation $\varepsilon\%D=100(CFD-EFD)/EFD$ and is positive for over prediction and negative for under prediction. The results showed a monotonic convergence for all the obtained variables, except for the 1\textsuperscript{st} amplitude of the computed resistance in wave.

| \(CT \times 10^3\) | \(z [\text{mm}]\) | \(\theta [\text{º}]\) |
|----------------|----------------|----------------|
| \(0^\text{th} \text{Amp.}\) | \(1^\text{st} \text{Amp.}\) | \(0^\text{th} \text{Amp.}\) | \(1^\text{st} \text{Amp.}\) | \(0^\text{th} \text{Amp.}\) | \(1^\text{st} \text{Amp.}\) |
| \(R_G\) | 0.894 | 1.003 | 0.655 | 0.444 | 0.5 | 0.5 |
| Status | M.C. | Div. | M.C. | M.C. | M.C. | M.C. |
| \(M_2 \%S_1\) | -0.360 | 0.884 | 1.983 | 1.284 | 1.563 | 0.511 |
| \(M_3 \%S_1\) | -0.763 | 1.765 | 5.011 | 4.180 | 4.688 | 1.532 |
| \(\delta_G\) | -0.0422 | N.A. | -0.057 | -0.411 | -0.002 | -0.008 |
| \(|\delta_G| \%D\) | 0.338 | N.A. | 1.588 | 2.070 | 1.095 | 0.472 |

M.C. = Monotonic Convergence, Div. = Divergence

The grid error in resistance for the 0\textsuperscript{th} amplitude is less than unity, while for the 1\textsuperscript{st} amplitude it could not be computed based on Richardson Extrapolation method due to the divergence. For the heave amplitudes, the grid error is relatively high, while for the pitch amplitudes, the error is within the unity value for the 0\textsuperscript{th} amplitude and less than unity for the 1\textsuperscript{st} amplitude.

The free-surface topology for Case 2 can be visualized in figure 7 for the four encounter quarters of the wave, as previously described. The interaction between hull and wave is more visualized here since the wave length is almost similar to the ship length. Two different wave systems are radiated by the fore and aft shoulders, causing a recognized disturbance in the initial wave system.

4.3. Case 3
Case 3 represents the long wave length condition as described before, where $\lambda_0=1.6L_{PP}$ with a longest wave period and smallest frequency. Simulation is performed for about 30 seconds. The Experimental data (EFD) were presented for this case in G2010 Workshop. Validation of CFD results is done by direct comparison with the EFD results. Table 7 includes the results for the 0\textsuperscript{th} and 1\textsuperscript{st} amplitudes for ship resistance, heave and pitch motions, while the time history of the ship resistance in waves is plotted in figure 8.

As previously concluded, all the obtained CFD results are showing a relatively reasonable agreement with the EFD data. The error for the computed resistance 0\textsuperscript{th} amplitude and the heave 1\textsuperscript{st} amplitude are the least obtained errors with 5.62% and 4.96%, respectively. Also, the other significant values comply with the average error range obtained in the G2010 [5].
Figure 7. Free-surface topology at different encounter instants for Case 2.

Table 7. Case 3 results compared to the EFD results [5].

|        | $C_T \times 10^3$ | z [mm] | $\theta$ [$^\circ$] |
|--------|-------------------|--------|---------------------|
|        | $0^{th}$ Amp.     | $1^{st}$ Amp. | $0^{th}$ Amp. | $1^{st}$ Amp. | $0^{th}$ Amp. | $1^{st}$ Amp. |
| EFD    | 8.256             | 79.433  | -3.593              | 26.905        | -0.155        | 2.554         |
| CFD    | 8.72              | 56.92   | -2.142              | 25.42         | -0.139        | 2.086         |
| $\epsilon\%D$ | 5.62$\%$ | -28.33$\%$ | -40.3$\%$ | -4.96$\%$ | -10.3$\%$ | -18.32$\%$ |

Figure 8. Time history for computed resistance in waves in Case 3.

The free-surface topology for Case 3 can be visualized in figure 9 for the four encounter quarters of the wave, as previously described. Similarly, the two wave systems are radiated by the fore and aft shoulders, causing a recognized disturbance in the initial wave system.

5. Added resistance in waves
The added resistance in waves can be simply approximated based on a direct comparison between the $0^{th}$ amplitude of the ship resistance computed in waves and the total resistance value obtained in calm water, according to equation 4 [13]

$$R_{aw} = \bar{R}_{aw} - R_{calm}$$  (4)
where $\bar{R}_{aw}$ is the average of the computed resistance in wave and $R_{calm}$ is the obtained or measured resistance in calm water. The calm water resistance value was presented in the G2010 Workshop, which was $C_{T\text{calm}}=5.14 \times 10^{-3}$, as measured in the INSEAN tank test [5]. The obtained $C_{Tw}$ values as presented in the previous sections are 7.7, 11.67 and $8.72 \times 10^{-3}$ for Cases 1, 2 and 3, respectively. The percentages of the added resistance for the three cases are about 49.8%, 127.04% and 69.65%, respectively. These values are considered significant and should be taken seriously into consideration.

For Case 2, the added resistance seems to be very significant, which rises a doubt about its value. However, the value is compared with the tank test. This might be due to the fact that this wave length is very close to the ship length, which is usually considered as a critical case and might be close to the resonance frequency of the ship. This requires extra investigation in the future.

6. Concluding remarks

The vertical motions and the added resistance in regular head waves of the KVLCC2 ship model were predicted numerically and presented in the current study, based on URANSE CFD approach. The results comprised computations for a single wave height and three different wave lengths, including long and short waves with respect to the ship length. Based on the aforementioned discussions in the previous sections of this paper and the presented results for ship resistance, heave, pitch, free-surface and added resistance in waves, the following points can be concluded:

- In Case 1, the computed results were compared with those obtained in G2010 Workshop based on five different commercial CFD codes, and showed a close agreement. This demonstrates the robustness and consistency of the numerical results obtained in this study.
- In Case 2, the results were within a moderate level of accuracy compared to the EFD data, though the errors for the 1st amplitude of the computed resistance and the 0th amplitudes of the motions were still high. The first conclusion is still applicable.
- The results for Case 3, from resistance and motions point of view, were also within a very close range with the results obtained in G2010 Workshop.
- The free surface topology was presented for the three cases, showing two different wave systems radiated by the ship fore and aft shoulders, which causes an interaction with the original wave system. The significance of this interaction was varying based on the wave length.
- The verification study for Case 2 showed a monotonic convergence for all the computed variables, except for the 1st amplitude of the resistance. In most of the computed results, the grid
error was less than the data uncertainties. A complete verification study is necessary to obtain the numerical uncertainties, which requires an extra time step convergence study.

- The added resistance in waves for the three cases was computed based on a simple difference approach between the obtained resistance coefficient in waves and the resistance value provided by the tank test for calm water case. The relative increment in resistance in wave compared to the one in calm water was 49.8%, 127.04% and 69.65% for Cases 1, 2 and 3, respectively. On the other hand, the value obtained for Case 2, though it had a good agreement with the EFD results, still raises a doubt and requires further investigation.

In general, the results obtained in this study, though they seem promising, further investigation is required for different wave lengths, heights and ship speeds. Besides, a thorough investigation for the non-linear phenomena, such as wave breaking, deck greenening, slamming loads, etc. should be taken into consideration in the future work.

7. References
[1] Tezdogan T, Demirel Y K, Kellett P, Khorasanchi M and Incicek A 2015 J. Ocean Eng. 97 186–206
[2] Deng G B, Queutey M and Visonneau M 2010 J. Hydrodynamics. 22(5) 476–481
[3] Sato Y, Miyata H and Sato T 1999 J. Marine Sc. & Tech. 4 108–116
[4] Carrica P M, Wilson R V, Noack R W and Stern F 2007 Comp. & Fluids, 36(9) 1415–1433
[5] Larsson L, Stern F and Visonneau M 2013 Numerical Ship Hydrodynamics, an Assessment of the Gothenburg 2010 Workshop (Netherlands: Springer)
[6] Sadat-Hosseini H, Wu P C and Carrica P M 2013. J. Ocean Eng. 59 240–273
[7] Kim M, Hizir O, Turan O and Incicrk A 2017 J. Ocean Eng. 140 466–476
[8] Park D M, Seo M G, Lee J, Yang K Y and Kim Y 2014 J. Soc. Nav. Archit. Korea 51(6) 459–479
[9] Kim H T, Kim J J, Choi N Y and Lee G H 2014 A study on the operating trim, shallow water and wave effect SNAK 631–637
[10] Kim Y C, Kim K S, Kim J, Kim Y S, van S H and Jang Y H 2015 Proc. 25th Int. Offshore & Polar Eng. Conf. (Hawaii: USA/ Int. Soc. Offshore Polar Eng.) 924–930
[11] Queutey P, Visonneau M, Leroyer A, Deng G and Guilmineau E 2008 Proc. 11th Num. Towing Tank Symp. (Brest: France) 11–16
[12] International Towing Tank Conference (ITTC) 2014 Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures Rev.2 7.5-03-01-01 1–12
[13] Ozdemir Y H and Barlas B 2017 Int. J. Naval Architecture & Ocean Eng. 9 149–159

Acknowledgment
The authors would like to acknowledge the multidisciplinary research platform “ReForm” of the “Dunarea de Jos” of Galati for the unfailing support in conducting the present research.