Numerical Investigation of the Shallow Water Effect on the Total Resistance, Vertical Motion and Wave Profile of a Container Ship Model

A Bekhit¹ and F Popescu²

¹“Dunarea de Jos” University of Galati, Faculty of Naval Architecture, 47 Domneasca Street, 800008, Galati, Romania
²“Dunarea de Jos” University of Galati, Faculty of Engineering, 47 Domneasca Street, 800008, Galati, Romania

E-mail: adham.bekhit@ugal.ro

Abstract. A ship sailing in shallow water is affected by the interaction between the moving hull and the seabed in different forms such as: significant increase in total resistance, increase in the sinkage and trim that may result in squatting effect, a change in wave pattern and increased wave amplitude, a change in the propeller wake field and an altered propeller and manoeuvring performance. In order to investigate the influence of shallow water on a container ship that is assumed to be subjected to work in different water depths, the KRISO container ship model is analysed in both deep and shallow water, with a special focus on the change in resistance, vertical motion and wave profile. A viscous flow simulation is performed first to predict the ship performance in deep water and the results are compared with the experimental data that are available in the public domain. Then, the critical shallow water condition is investigated and the obtained results are compared against those formerly obtained in deep water. The numerical simulations are performed using the viscous flow solver ISIS-CFD of the FINE™/Marine software provided by NUMECA. The solver is based on the finite volume method to build the spatial discretization of the transport equation in order to solve the Reynolds-Averaged Navier-Stokes (RANS) equations. Closure to turbulence is achieved using the Menter Shear Stress Transport (K-ω SST) model, while the free-surface is captured through an air-water interface based on the Volume of Fluid (VOF) method. The comparison between the numerically obtained results and the available EFD data showed a satisfactory congruence.

1. Introduction

The maritime transport represents one of the most important factors in the world trade activities. Over 80 per cent of global trade by volume and more than 70 per cent of its value being carried on board ships, as reported in the Review of Maritime Transport 2017 [1]. The inland water transport can enhance significantly the quality and the amount of transported goods with a cheaper, more safe and effective transportation mode. The only drawbacks may be regarding the fear of pollution hazards and shallow water problems, that may tend to cause damage to the ship and its propulsion system. For this reason, and in order to reduce the grounding incidents of the ship it is very important to study the ship performance in shallow water. A ship sailing in shallow water is affected by the interaction between the moving hull and the seabed in different forms such as: significant increase in total resistance, increase in the sinkage and trim that may result in squatting effect, a change in wave pattern and
increased wave amplitude, a change in the propeller wake field and an altered propeller and manoeuvring performance. For this purpose, studying the shallow water effect on a ship is crucially important to provide sufficient powering for those ships that were especially designed for working in shallow waters and coastal zones where the water depth is limited.

In such complicated circumstances, the experimental approach is inconvenient and not practicable from cost and risk points of view. The best alternative is to use the Computational Fluid Dynamics (CFD) to investigate the ship performance in shallow water, as it provides a significant flexibility for these conditions and the obtained results are highly reliable, especially with the significant development recorded in the past two decades in physical modelling and computational power.

Heading form this perspective, this study investigates the shallow water effect on a container ship that is assumed to be sailing in shallow water condition with different water depth based on a CFD viscous flow method. Six different cases with respect to the water depth including six different ship velocities in each case were investigated with special focus on the ship resistance, vertical motions and free-surface flow. In order to insure the accuracy of the computed results, a validation test against the available Experimental Fluid Dynamics (EFD) data was performed especially for the computed total resistance and showed a proper agreement with the available data.

2. Geometry and analysis conditions
The current study is applied on the ship model of the Korea Research Institute of Ships and Ocean Engineering (KRISO) Container Ship (KCS). The ship was conceived to provide benchmark data for both explication of flow physics and CFD validation for a modern container hull 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 with different scopes to predict ship resistance, free-surface, local flow configurations, seakeeping and maneuverability. The results for these tests can be found in the public domain for CFD verification and validation purposes.

In this case study, the ship model of scale 1/31.6 is used for validating this simulation, which has a length between perpendiculars of \( L_{pp} = 7.2786\) m and a draft of 0.3418m. Full details of the ship characteristics are tabulated in table 1 for both model and full scale, while the geometry of the ship is depicted in figure 1.

| Parameter                  | Unit | Full Scale | Model Scale |
|----------------------------|------|------------|-------------|
| Scale, \( \lambda \)      | -    | 1          | 31.6        |
| Length, \( L_{pp} \)      | [m]  | 230.0      | 7.2786      |
| Beam, \( B \)             | [m]  | 32.2       | 1.0190      |
| Draft, \( T \)            | [m]  | 10.8       | 0.3418      |
| Depth, \( D \)            | [m]  | 19.0       | 0.6013      |
| Block coefficient, \( C_B \)| -   | 0.6505     | 0.6505      |
| Mid-ship section coefficient, \( C_M \)| -   | 0.9849     | 0.9849      |
| Displacement of volume, \( \rho \) | [m³] | 52030     | 1.6490      |
| Wetted surface area w/o rudder, \( S_0 \) | [m²] | 9424     | 9.4379      |

Figure 1. KCS ship model geometry.
The analysis conditions include six different cases; every case consists of six velocities corresponding to Froude number \((Fr = \frac{V}{\sqrt{gL_{PP}}}\) range from \(Fr=0.108\) to \(0.282\), where \(V\) represents the ship velocity and \(g\) is the gravitational acceleration and \(L_{PP}\) stands for the ship length. In table 2, the simulation cases are categorized with respect to the ship velocity, the corresponding Froude number, water depth \(H\), and water depth to draft ratio \(H/T\).

**Table 2. Simulation cases and corresponding ship velocity.**

| Ship Velocity | Water Depth \((H)\)               | V\([\text{m/s}]\) | Fr\([-\) | Water Depth \((H)\)               |
|--------------|---------------------------------|------------------|----------|---------------------------------|
| 1.5 \(L_{PP}\) | 10.92 m H=10.92 m \(H/T=35\)     | 0.92      | 0.108   | C1.1                            |
| 1.0 \(L_{PP}\) | 7.28 m H=7.28 m \(H/T=15.95\)     | 1.28      | 0.152   | C1.2                            |
| 0.5 \(L_{PP}\) | 3.64 m H=3.64 m \(H/T=10.65\)     | 1.65      | 0.195   | C1.3                            |
| 0.25 \(L_{PP}\) | 1.82 m H=1.82 m \(H/T=5.32\)     | 1.92      | 0.227   | C1.4                            |
| 0.125 \(L_{PP}\) | 0.91 m H=0.91 m \(H/T=2.66\)     | 2.20      | 0.266   | C1.5                            |
| 0.0625 \(L_{PP}\) | 0.455 m H=0.455 m \(H/T=1.33\)     | 2.38      | 0.282   | C1.6                            |

\(C=\)Simulation Case Number

3. Numerical Simulation Framework

3.1. Numerical solver

The numerical simulations performed in this study are carried out using the ISIS-CFD solver of the software FINE\(^\text{TM}\)/Marine provided by the NUMECA. The solver is based on the finite volume method to build the spatial discretization of the transport equation to solve the Reynolds-Averaged Navier Stokes Equations (RANSE) [2]. The spatial discretization is face-based in which the fluxes are constructed face by face, which means that any arbitrary shape control volumes with arbitrary number of faces can be used. Hence, it is suitable for the use of unstructured grids to provide more flexibility for discretising complex hull geometries, ship appendages, etc. The temporal discretization is cell-centred based on a second order three-level scheme. The velocity field is obtained from the momentum equation; while the pressure is extracted from the mass constraint or continuity equation transformed into a pressure equation conform a velocity pressure coupling based on a Rhie and Chaw SIMPLE type algorithm [3]. Closure to turbulence is achieved through the blended Menter’s Shear Stress Transport \(K-\omega\) SST turbulence model. Multi-phase flow is used to model the free-surface interface based on the Volume of Fluid (VOF) interface capturing technique, where incompressible and non-miscible flow phases are modelled by introducing conservation equations of the volume fraction for each volume phase/fluid [2]. Convection and diffusion terms in the governing equations are discretized using second-order upwind and central differencing scheme, respectively.

3.2. Governing equations

The time averaged continuity and momentum equations for the incompressible flow with external forces, can be written in tensor form, in the Cartesian coordinate system as:

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

\[
\frac{\partial (\rho \bar{u}_j)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_i \bar{u}_j + \rho \bar{u}'_i \bar{u}'_j \right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}
\]
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

$$\bar{\tau}_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (3)$$

3.3. Computational domain and boundary conditions

The computational domain is having a rectangular prism configuration as schematically presented in figure 2. The main dimensions of the computational domain in $(x-y-z)$ direction are $(5.0 L_{pp} - 2.0 L_{pp} - (0.5 + h) L_{pp})$, where $H$ is the depth factor which varies based on the assumed water depth such that $(H=1.5, 1.0, 0.5, 0.25, 0.125 L_{pp}$ and $0.0625 L_{pp}$, as illustrated in figure 2. The inflow boundary is located at $2.0 L_{pp}$ upstream, the exit boundary is located at $3.0 L_{pp}$ downstream, $2.0 L_{pp}$ between the ship symmetry plan and the side boundary, the top boundary is located at $0.5 L_{pp}$ above the undisturbed free-surface level, which is located at $z=0.3148$ 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, 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.

![Figure 2. Computational domain, dimensions and boundary conditions.](image)
is tabulated in table 3, while figure 3 shows the computational grid for the case number C3, where $H$ is $0.5L_{pp}$. Two levels local refinements are applied on the free-surface by imposing a global surface refinement at the undisturbed water level for the entire domain with relatively coarser grid, while a dense grid is applied on the Kelvin pattern zone with a wider opening angle of 80º for the shallow water cases to capture more details of the generated wave by the ship, as shown in figure 3.c. Nevertheless, for case 1, the Kelvin angle is retained at 20º. The refinement for the Kelvin pattern is chosen to provide 60 grid cells per wave length in $x$- and $y$-directions such that $\Delta x=\Delta y=2\pi Fr^2L_{wl}/60$; while in $z$-direction, the thickness of the refinement zone is chosen to cover the stagnation wave amplitude defined by $\zeta_{max}=0.5U^2/g$. The cell size in $z$-direction for the free-surface refinement sector is chosen as $\Delta z= L_{wl}/1000$. The bottom zone is divided in two sections, both sections are treated with wall modelled approach having the first distance to the wall corresponding to $y^*=60$. The first section which is located underneath the hull is treated with a thicker viscous layer with 8 levels, while only 4 viscous layers were imposed for the downstream section. The hull is also treated with a wall-modelled approach having $y^*=60$.

**Table 3. Number of grid cells for simulation cases.**

| Number of grid cells (M) | Water Depth ($H$) |
|--------------------------|-------------------|
|                          | 1.5$L_{pp}$ | 1.0$L_{pp}$ | 0.5$L_{pp}$ | 0.25$L_{pp}$ | 0.125$L_{pp}$ | 0.0625$L_{pp}$ |
|                          | 4.188      | 5.847      | 10.487     | 17.355      | 18.196       | 18.686        |

**Figure 3.** Discretization grids showing: (a) 3D view, (b) side view and (c) Top view.

### 3.5. Solution strategy and available resources

All the simulations are performed with the help of a High Performance Computing (HPC) machine with available 120 cores at 2.5 up to 3.3 GHz. For resistance prediction, the simulation is performed for 30 seconds to ensure adequate numerical convergence for resistance and vertical motions. The flow is accelerated based on a steady quasi-static approach for a selected period based on the ship speed to satisfy the condition $T_{acc} = 2L_{pp}/U$ in order to avoid any numerical instabilities in the beginning of the simulation. 10 non-linear iterations are used with a time step $\Delta t$ is chosen based on the ITTC formula for RANSE simulations with two-equations turbulence models as $\Delta t = 0.005 L_{pp}/U$ [3,4].
4. Results and Discussions

4.1. Resistance Results

In the scope of validating the computed results against the available EFD data retrieved from [5,6], the computed total resistance coefficient is represented against the EFD in table 4 for case number C1 where the water depth is set at $H=1.5L_{PP}$. The obtained results showed a good agreement with the EFD data with an absolute error range between 0.35 and 2.0%. This can highlight the accuracy of the computed results.

| Ship Velocity | $C_T \times 10^4$ | Error% $C_T$ [-] |
|---------------|-------------------|------------------|
| V [m/s]       | $Fr$ [-]          | EFD              | CFD              | $\epsilon\% = \frac{100(CFD - EFD)}{EFD}$ |
| 0.92          | 0.108             | 3.796            | 3.723            | -1.925 |
| 1.28          | 0.152             | 3.641            | 3.604            | -1.006 |
| 1.65          | 0.195             | 3.475            | 3.463            | -0.357 |
| 1.92          | 0.227             | 3.467            | 3.496            | 0.829 |
| 2.20          | 0.26              | 3.711            | 3.675            | -0.960 |
| 2.38          | 0.282             | 4.501            | 4.463            | -0.850 |

Based on the obtained results the comparison between the different cases with respect to the water depth is listed in table 5 showing the effect of the shallow water on the increment in ship resistance, while figure 4 shows the bar chart diagram for the ship resistance based on the water depth. The theoretical principle remains valid for the computed results, since the ship resistance increases as the water depth decreases. The only exception is between the three cases where $H=1.5L_{PP}$, $H=1.0L_{PP}$ and $H=0.5L_{PP}$ where the values remain within a certain level even for the high speed. This might be related to the fact that $H/T$ ratio is reasonable and the change is insignificant in comparison with the water depth; yet, it might be an influence of the numerical errors that result in this fluctuation. This might be verified in the future with a grid convergence study.

| Ship Velocity | $H=1.5L_{PP}$ | $H=1.0L_{PP}$ | $H=0.5L_{PP}$ | $H=0.25L_{PP}$ | $H=0.125L_{PP}$ | $H=0.0625L_{PP}$ |
|---------------|--------------|--------------|---------------|----------------|----------------|-----------------|
| V [m/s]       |              |              |               |                |                |                 |
| 0.92          | 15.042       | 15.106       | 15.084        | 16.124         | 16.762         | 21.404          |
| 1.28          | 28.19        | 28.356       | 28.52         | 31.944         | 32.206         | 35.438          |
| 1.65          | 45           | 44.912       | 44.822        | 47.694         | 48.102         |                 |
| 1.92          | 61.516       | 61.398       | 60.958        | 70.114         | 73.862         |                 |
| 2.20          | 84.916       | 84.936       | 84.154        | 106.224        | 119.926        |                 |
| 2.38          | 120.67       | 120.626      | 120.602       | 138.620        | 144.962        |                 |

Figure 4. Resistance comparison based on the water depth
4.2. Vertical motions

The sinkage and trim for the six cases were computed and represented in table 6 and 7 showing the fact that the ship sinkage and trim increases with the ship speed in shallower water due to the speed effect which tends to create a low pressure zone that consequently results in a suction force that pulls the ship bottom towards the seabed. It is worth mentioning that there some values in the shallower water that do not comply with this criteria, this might be related to iterative errors as the fluctuation of the computed values increases significantly as the ship speed increases. The values recorded represent an average for the last 10% of the simulation time that lasted for 60 seconds. The authors think that this time should be increased, especially for the shallowest cases to enhance the convergence criteria.

| Ship Velocity [m/s] | \( H=1.5L_{pp} \) | \( H=1.0L_{pp} \) | \( H=0.5L_{pp} \) | \( H=0.25L_{pp} \) | \( H=0.125L_{pp} \) | \( H=0.0625L_{pp} \) |
|---------------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| 0.92                | 0.351           | 0.350           | 0.352           | 0.297           | 0.295           | 0.325           |
| 1.28                | 0.349           | 0.348           | 0.353           | 0.313           | 0.313           | 0.341           |
| 1.65                | 0.346           | 0.347           | 0.346           | 0.332           | 0.332           | 0.328           |
| 1.92                | 0.343           | 0.342           | 0.342           | 0.335           | 0.335           | 0.337           |
| 2.20                | 0.339           | 0.338           | 0.338           | 0.341           | 0.341           | 0.344           |
| 2.38                | 0.335           | 0.335           | 0.335           | 0.346           | 0.346           | 0.349           |

4.3. Free-surface

It is well known that the ship sailing in shallow water encounters a significant effect on the free-surface topology that results in changing the Kelvin wave pattern. For this purpose, the wave pattern in shallow water is represented in this section only for water depths \( H=0.5L_{pp} \), \( H=0.125L_{pp} \) and \( H=0.0625L_{pp} \). A special form of the computed dated based on the water depth of \( H=0.25L_{pp} \) was also investigated for higher ship speeds up to the squatting condition to investigate the change in wave profile correspondingly.

4.3.1. Case C3 with \( H=0.5L_{pp} \). The free-surface topologies for the cases C3.3, C3.4, C3.5 and C3.6 which are corresponding to the ship velocity of \( V=1.65 \), 1.92, 2.20 and 2.38, respectively are depicted in figure 5. The Kelvin pattern seems to not be affected even for the high speed ship, which actually supports the previously presented results where it was showing that for the water depth of \( H=0.5L_{pp} \) the effect of depth is insignificant and gives similar values for resistance and vertical motions like the \( H=1.5L_{pp} \) case.

4.3.2. Case C5 with \( H=0.125L_{pp} \). The corresponding free-surface topology for this case can be visualized in figure 6, where the effect of the increased ship speed results in a noticeable deformation in the Kelvin wave pattern envelope angle, which tends to increase as the ship speed increases. This complies positively with the theoretical principles regarding the shallow water effect on free-surface.

| Ship Velocity [m/s] | \( H=1.5L_{pp} \) | \( H=1.0L_{pp} \) | \( H=0.5L_{pp} \) | \( H=0.25L_{pp} \) | \( H=0.125L_{pp} \) | \( H=0.0625L_{pp} \) |
|---------------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| 0.92                | 0.022           | 0.021           | 0.022           | 0.023           | 0.024           | 0.050           |
| 1.28                | 0.048           | 0.049           | 0.049           | 0.055           | 0.060           | 0.151           |
| 1.65                | 0.090           | 0.087           | 0.088           | 0.111           | 0.124           | Ship Grounded   |
| 1.92                | 0.128           | 0.128           | 0.128           | 0.172           | 0.196           |                |
| 2.20                | 0.156           | 0.157           | 0.157           | 0.232           | 0.285           |                |
| 2.38                | 0.172           | 0.174           | 0.175           | 0.261           | 0.383           |                |
Figure 5. free-surface profile for case C3 with $H=0.5L_{pp}$. 

\[ V = 1.65 \text{ m/s} \]

\[ V = 1.92 \text{ m/s} \]

\[ V = 2.20 \text{ m/s} \]

\[ V = 2.38 \text{ m/s} \]

\[ V = 1.28 \text{ m/s} \]

\[ V = 1.65 \text{ m/s} \]
4.3.3. Case C6 with $H=0.0625L_{pp}$. The free-surface topology for case C6 is depicted for two ship speeds in figure 7. The same comments that were introduced for case C5 remains also valid for this case.

4.3.4. Case C7* with $H=0.1L_{pp}$. In order to highlight the significance of ship velocity on the wave pattern for even higher speeds, an extra simulation case on a coarser grid with 3.415M cells was introduced especially after the grounding of the case C6 for the higher set of speeds. Another water depth was selected between cases C5 and C6 having the water depth within $H=0.1L_{pp}$ for this purpose two extra ship velocities were calculated which are $V=3.17$ and $4.12$ m/s which are close to the critical squatting condition corresponding to Froude number based on depth close to unity. The deformation in the wave pattern is significant and the opening angle of Kelvin pattern is increased dramatically. Also wave profile and wave crests and troughs distribution seem to be highly affected.

5. Conclusions
The shallow water effect on the KCS benchmark ship model based on a CFD viscous flow solver was presented with respect to the influence of the water depth on the total ship resistance, the vertical
motions and finally the wave profile. Six different simulation cases assigned based on the water depth with six different ship speeds were analysed and investigated individually for every condition, in order to highlight their influence on the obtained results. In the scope of validating the numerically obtained results, a direct comparison validation test was performed against the EFD data showing a good agreement between CFD and EFD results with an error less than 2%.

\[
V = 1.92 \text{ m/s} \\
V = 2.20 \text{ m/s} \\
V = 3.17 \text{ m/s} \\
V = 4.12 \text{ m/s}
\]

**Figure 8.** Free-surface profile for case C7* with \(H=0.1L_{PP}\).

The total ship resistance results for every case were compared to the deep water outcomes showing the increment in total resistance with respect to the reduction of water depth. It was concluded that until the water depth of \(H=0.5L_{PP}\) the effect of depth was still insignificant resulting in almost similar resistance values.

For the vertical motions, the sinkage and trim were computed and showed to comply with the theoretical approach for the shallow water effect except for some discrepancies that resulted from the iterative error. This might be important to increase the total simulation time, especially for the shallowest water cases to improve the convergence criteria.

Free-surface results showed to have a proper agreement with the theoretical shallow water effect for all the simulated and presented results. It may be concluded that the grid density for the coarsest grids could be enhanced to capture more accurately the wave profile for the smallest ship speeds.
The future plan is to investigate the influence of the shallow water on the wake flow characteristics, propulsion efficiency and the manoeuvrability of the ship.

6. References
[1] United Nations Conference on Trade and Development (UNCTAD) 2017 Review of Maritime Transport 2017, United Nation, New York and Geneva.
[2] Deng G B, Queutey M and Visonneau M 2010 J. Hydrodynamics. 22(5) 476–481.
[3] Queutey P, Deng G B, Wackers J, Guilmineau E, Leroyer A and Visonneau M 2012 J. Ship Technology Research 59(2) 44-58.
[4] International Towing Tank Conference (ITTC) 2011, Practical guidelines for ship CFD applications. Rev.1 7.5-03-02-03 1–18.
[5] Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics https://t2015.nmri.go.jp/ accessed on 15.02.2021.
[6] Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics https://t2015.nmri.go.jp/Presentations/Day2-AM2-KCS-Resistance_SP-Kim.pdf accessed on 15.02.2021.

Acknowledgement
This work was carried out in the framework of the research project DREAM (Dynamics of the RESources and technological Advance in harvesting Marine renewable energy), supported by the Romanian Executive Agency for Higher Education, Research, Development and Innovation Funding – UEFISCDI, grant number PN-III-P4-ID-PCE-2020-0008.