CFD Prediction of Ship-Bank Interaction

A Lungu

“Dunarea de Jos” University of Galati, 47 Domneasca Street, Galati 800008, Romania
adrian.lungu@ugal.ro

Abstract. Ships operating in different shallow-water areas are at risk of grounding due to squat, heel and wave-induced motions. The paper aims not only at assessing the ship squat in any bathymetry, but also to reveal the waves influence on the environment. A 3D numerical model for predicting ship resistance in confined waters by taking the ship sinkage into account is proposed. A time-domain model is used to predict unsteady squat and dynamic acceleration effects for a vessel travelling in non-uniform water depth. A quasi-steady approach where the prediction at each time step is based on steady-state heave force and pitch moment in uniform water depth is used. Resistance, sinkage and trim, the free-surface elevation as well as the wake flow structure are considered in the ship performance analysis. The results show the wave pattern in the subcritical, critical and supercritical regions. The total resistance/drag is calculated at various ship speeds in shallow water using CFD and compared with the calculated deep water resistance. Attention is paid to the verification and validation based on a grid convergence study, as to the exploration of the modelling error in RANS computations to enable more accurate and reliable predictions of the bank effects.

1. Introduction
The effect of shallow water and restricted waterways on the resistance of ships has increased in the recent years because of the growth in ship size and the increased congestion of the shipping routes. When a ship moves in a restricted waterway a significant flow contraction occurs around the hull. Consequently, the flow velocity under the hull augments and a pressure drop similar to the Bernoulli Effect takes therefore place, which is finally causing a loss of buoyancy. The pressure drop manifests in a vertical move of the hull, which is called sinkage.

The two phenomena have a significant influence on the hydrodynamic ship resistance: the accelerated flow leads to a growth of hull friction, whereas the sinkage determines an augmentation of the water mass that the ship has to push when moving ahead. Other effects of the navigation in restricted waterways are represented not only by the increase of the wave train amplitudes which eventually leads to a prominent erosion process of the banks, but also a change of the free-surface topology. Because of the interaction between the propagating waves and the boundaries of the channel, an interference phenomenon occurs and the classic Kelvin wave cannot be recovered anymore.

The numerical prediction of an inland water ship resistance in shallow water is therefore important for the accurate estimation of the onboard power design requirements [1]. The produced hydrodynamic forces in shallow water may considerably affect the manoeuvring performance of the ship, making it difficult to steer. The ship may collide with the side wall or run aground due to the so-called squat phenomenon. Despite its theoretical and practical importance, only a few attempts have been made in the past decades to describe these interactions by using exclusively computational fluid dynamics (CFD
hereafter) estimations. Given their achieved maturity so far, the CFD methods may seemingly account in the near future as reliable and accurate computation of the interaction forces in this particular navigation case.

To the knowledge of the author, the pioneering work proposed in [2] in which the unsteady Chimera RANS method was employed represents a milestone in studying the ship-to-ship interaction in shallow and restricted waterways. A validation of the method against some available experimental measurements provided in the referenced work has to be mentioned as well. As reported by Chen et al., the computed interaction forces and moments matched very well with the experimental results. They also investigated the importance of including the free surface, sinkage and trim and the influence of wall boundary conditions. A few years later, Lo et al. applied the FLOW-3D CFD commercial software to simulate the bank effect on the KCS 3600 TEU container ship model in [3]. Their work proved that CFD techniques achieved a sufficient accuracy to simulate reliably bank effects without the need for conducting extensive model tests.

Zou et al. [4, 5] used the CFD techniques to investigate bank effects on a tanker moving straight ahead at low speed in a canal characterized by surface-piercing banks. The reported work included grid convergence studies as well as verification and validation tests based on comparisons with experimental data. For varying water depths and ship-to-bank distances, the sinkage and trim as well as the wall boundary conditions. They also investigated the importance of including the free surface, sinkage and trim and the influence of wall boundary conditions. A few years later, Lo et al. applied the FLOW-3D CFD commercial software to simulate the bank effect on the KCS 3600 TEU container ship model in [3]. Their work proved that CFD techniques achieved a sufficient accuracy to simulate reliably bank effects without the need for conducting extensive model tests.

To the knowledge of the author, the pioneering work proposed in [2] in which the unsteady Chimera RANS method was employed represents a milestone in studying the ship-to-ship interaction in shallow and restricted waterways. A validation of the method against some available experimental measurements provided in the referenced work has to be mentioned as well. As reported by Chen et al., the computed interaction forces and moments matched very well with the experimental results. They also investigated the importance of including the free surface, sinkage and trim and the influence of wall boundary conditions. A few years later, Lo et al. applied the FLOW-3D CFD commercial software to simulate the bank effect on the KCS 3600 TEU container ship model in [3]. Their work proved that CFD techniques achieved a sufficient accuracy to simulate reliably bank effects without the need for conducting extensive model tests.

Zou et al. [4, 5] used the CFD techniques to investigate bank effects on a tanker moving straight ahead at low speed in a canal characterized by surface-piercing banks. The reported work included grid convergence studies as well as verification and validation tests based on comparisons with experimental data. For varying water depths and ship-to-bank distances, the sinkage and trim as well as the wall boundary conditions. They also investigated the importance of including the free surface, sinkage and trim and the influence of wall boundary conditions. A few years later, Lo et al. applied the FLOW-3D CFD commercial software to simulate the bank effect on the KCS 3600 TEU container ship model in [3]. Their work proved that CFD techniques achieved a sufficient accuracy to simulate reliably bank effects without the need for conducting extensive model tests.

An exhaustive study on the ship–bank interaction effects in which experiments and different solvers were used to predict the hydrodynamic forces and moments on the ship was recently reported in [6]. The considered ship hull model was the KVLCC2 for which extensive experimental and numerical investigations were performed. Experiments conducted on a KVLCC2 model of the have been reproduced numerically by making use of the viscous flow solvers REFRESCO and ISIS-CFD and the potential flow solver ROPES for validation purposes.

The present research follows a previous work of the author [7], which dealt with the numerical simulation of the squatting phenomenon performed for the KCS model hull. For the sake of consistency of the comparisons, the investigations performed herein employ not only the same ship hull, but also the same numerical approach. The research reported here is performed for a series of four water depths and three canal widths. Numerical solutions are compared to the experimental data to emphasize the influence of the restricted waterway on the ship total resistance.

2. Numerical approach

The ISIS-CFD solver, a part of the FINE™/Marine software package, is employed in the present study for computing the numerical solution. The unsteady RANSE ISIS-CFD solver built on the finite volume technique is suited to marine hydrodynamics applications. The anisotropic two equation Explicit Algebraic Stress Model (EASM) is employed for attaining the closure to turbulence.

The finite volume method is used to build the spatial discretization of the transport equations. The unstructured discretization is a face-based one in which all the unknown variables are cell-centered and the equations sets used in the implicit time stepping procedure are constructed face by face. Fluxes are computed in a loop over the faces and the contribution of each face is then added to the two cells next to a given face. Since this approach does not require any specific requirements on the cells topology, grids are fully unstructured, containing cells with arbitrary numbers of faces of different geometries. Pressure-velocity coupling is enforced through a Rhie and Chow SIMPLE type technique in which the velocity field is extracted from the momentum equations and the pressure is found from the mass conservation law, which is transformed into a pressure equation at each time step. The second order accurate Picard’s procedure is used for the linearization of the equations.

Free-surface position is predicted with a multi-phase flow approach in which the water surface is captured with a conservation equation for the volume fraction of water, discretized with a specific compressive numerical scheme. Unsteady computations are carried out in all the considered cases, as it
will be described further on in the following sections. Resistance, sinkage and trim, the free-surface elevation as well as the wake flow structure are taken into account in the ship performance analysis.

All the boundary conditions are either Dirichlet or Neumann on the open surfaces, the symmetry is imposed on the center line, while the no-slip condition is used on all the solid boundaries, as shown in Figure 1 which bears out the computational domain and boundary conditions formulation. At the upstream the flow starts from the still state and it is accelerated during the first 6 seconds according to a half-sinusoidal ramp. For all the solid walls the wall function is imposed as boundary condition.

The time step size is determined such that the Courant–Friedrichs–Lewy number or simply Courant number, which is the ratio of the physical time step $\Delta t$ to the mesh convection time scale, $\text{CFL} = \frac{U_\infty \cdot \Delta t}{\Delta x}$ is kept under close control. The CFL is typically calculated for each cell and kept at a value less than or equal to unity for numerical stability in all the computations reported here. All the computations reported in the present paper were performed on a parallel basis on 120 cores.

3. Hull geometry and computational grids

Assuming that $L_{pp}$ is the length between perpendiculars of the considered hull, the computational domain of the present numerical study extends $1.5 \cdot L_{pp}$ at the upstream of the hull and $3 \cdot L_{pp}$ at the downstream of it. On the lateral direction three different distances of the bank measured from the centreline are considered, $B/L_{pp} = 0.8, 0.4$ and $0.2$. On the vertical direction, a distance of one and a half $L_{pp}$ is chosen for the air domain in all the reported computations. As far as the water domain below the free-surface is concerned, $2 \cdot L_{pp}$ are chosen for the unlimited depth computational case, whereas $H/T = 2.5, 2.0, 1.5$ and $1.25$ are considered. $H$ represents the water depth measured below the undisturbed free-surface, whereas $T$ is the ship draught in the calm water condition.

The hull depicted in Figure 2 (a) is appended with a suspended rudder whose wetted area is 0.0801 m$^2$. The model length between perpendiculars is 7.2886 m, according to the 2.1 Case in [8]. The KCS model has a significant forecastle at the fore extremity and a planar deck in the rest of the hull. The main
particulars of the ship that correspond to the Case 2.10 in [8] are as follows: length between perpendiculars is \( L_{pp} = 7.2786 \) m, whereas the maximum beam of the waterline is \( B_{WL} = 1.019 \) m, draft \( T = 0.3418 \) m, displacement volume \( V = 1.649 \) m\(^3\), block coefficient \( C_B = 0.651 \), wetted area without rudder \( S_W = 9.4379 \) m\(^2\) and the wetted area of the suspended rudder is \( S_R = 0.1152 \) m\(^2\).

The computational unstructured meshes needed for the present study are generated in HEXPRESS. Volumes of a higher cell density are placed a-priori wherever the gradients of the computed physical parameters are expected to occur. Aside the mesh clustering, special attention was paid for fulfilling the minimal requirements concerning the smoothness and orthogonality. A viscous layer consisting on 28 cells whose minimum thickness is \( 10^{-6} \) m is introduced to keep the \( y^+ \) below unity.

### 4. Results and discussions

#### 4.1. Grid convergence test

Following the methodology described in [9] and [10], four different meshes were generated for the grid convergence test. Let these grids be denoted by G1 for the coarsest and G4 for the finest one. The G4 grid is depicted in Figs. 2(b) and 2(c) in which for reasons related to the clarity of the representation, every fourth grid line is drawn. The cell numbers of the generated meshes expressed in million cells are: 4.97, 10.02, 20.03 and 40.58, respectively. For all the grids additional requirements regarding the clustering inside the solid boundary layers, i.e. the ship hull and the channel vertical walls and bottom are imposed in the generation process. The grid convergence test is performed for three Froude numbers ranging from 0.152 to 0.227 as tabulated in Table 1 in which the computed solutions for the unbounded flow are compared to the experimental data (EFD) reported in [8].

| Fr       | EFD       | CFD G1 | CFD G2 | CFD G3 | CFD G4 |
|----------|-----------|--------|--------|--------|--------|
|          | \( Fr = 0.152 \) | \( Fr = 0.195 \) | \( Fr = 0.227 \) |
| EFD      | 3.641     | 3.475  | 3.467  |
| G1 \( C_T \times 10^3 \) | 3.778 | 3.362  | 3.359  |
| \( \varepsilon \% \) | 3.76  | 3.25   | 3.12   |
| G2 \( C_T \times 10^3 \) | 3.507 | 3.368  | 3.572  |
| \( \varepsilon \% \) | 3.68  | 3.08   | 3.03   |
| G3 \( C_T \times 10^3 \) | 3.758 | 3.569  | 3.532  |
| \( \varepsilon \% \) | 3.21  | 2.71   | 1.87   |
| G4 \( C_T \times 10^3 \) | 3.743 | 3.518  | 3.501  |
| \( \varepsilon \% \) | 2.80  | 1.24   | 0.98   |

The graphic representation of the absolute error of the numerical solution is proposed in Figure 3 in which \( N_C \) signifies the number of cells. Since the grid is mainly unstructured, the dependence of the numerical error on the number of cells is somehow difficult, therefore on the abscissa Figure 3 is considered the cubic root of \( N_C \). Regardless the \( Fr \) number value, the level of the error decreases monotonically with the mesh accuracy, a fact which confirms the grid convergence. Because the verification and validation calculation (V&V) was performed in [9] and [10] on the same hull and the same running conditions, for reasons related to the limited space, it will be therefore skipped here.

#### 4.2. Numerical solutions

The fineness of the discretization may be reflected not only by the grid convergence test, but also by the \( y^+ \) distribution on the hull, knowing its crucial importance on the accuracy of the pressure field within the boundary layer. Except for the area around the bilge keel downstream of the fore hydrodynamic shoulder, and a small region on the upper part of the rudder, where \( y^+ \approx 0.50 \), in all the rest of the domain its value does not exceed 0.4, as Figure 4 bears out. That may explain the low level of errors of the numerical solutions in respect to the EFD [8] tabulated in Table 1. Similar findings are reported in [11].
As mentioned above, the numerical simulation of the flow around the ship hull moving in the channel will be carried out for a set of three breadths and for four water depths. Six different Froude numbers, varying progressively from \( Fr = 0.155 \) to \( Fr = 0.205 \) will be considered in this study. Obviously, the effect induced by the banks on the main hydrodynamic parameters of the flow should be studied based not only on the reference solution for the unbounded domain, but also on the solutions computed for the navigation in shallow water previously reported by the author in [7]. In this respect, Figure 5 shows the free-surface profile computed for the unbounded flow at \( Fr=0.165 \). As it may be seen in the figure, the Kelvin angle for the wave train seems to be well reproduced.

Three wave sub-systems can be identified. That is, two divergent waves of a significant amplitude, originating from the fore and the stern, cohabitate and eventually interfere far downstream of the hull.
A transversal wave system of a lower magnitude propagates behind the hull and slightly vanishes in the flow, obviously due to the viscous dissipation. A significant wave crests is developed around the bow, as expected, which may be susceptible to lead to the occurrence of a sort of sub-breaking phenomenon if the velocity exceeds a critical value, as described in detail in [12]. In a sense, a Froude number of 0.165 might be considered as being rather high for the navigation in a limited environment since for the ship at the full size leads to a speed of 15.4 knots. However, given the importance of direct effect of a too high speed navigation on the environment, in the followings that condition will be considered in most of the cases aimed at emphasizing its effect on the restricted working conditions.

Computations carried out for the three channel widths have shown that when the distance between the ship hull and the bank decreases, lower pressures are generated on both sides of the ship and therefore the suction force towards the banks increases. The lower pressure area leads finally to an increase of the forces and moments acting on the hull. Because the channel width proved to play an important role on the wave amplitudes projected on the banks, only the most restrictive case of $B/\ell_{pp} = 0.2$ shown in Figure 6 will be considered in the followings since that case may be regarded as the most critical regime of navigation. To ensure that due to sinkage, the cells below the hull are not compressed too much (which may result in negative cell volumes), an adaptive grid refinement is used starting from a steady solution obtained on an initial mesh. A combined refinement criteria for both the free-surface and the pressure hessian is used [13]. The minimum size limit for refined cells has been imposed and activated every 50 steps of the time marching procedure, which finally led to an increase of the grid with about 20 million additional cells.

Figure 6. Free surface profile of the restricted flow computed for $Fr=0.165$, $H/T = 1.25$, $B/\ell_{pp} = 0.2$.

Figure 7 proposes a comparison between the free-surface profiles computed for four different relative depths at $Fr=0.195$. For the deepest water case considered in the present study, the wave making process affected by the proximity of the bottom and banks determines a rather significant change of the free-surface topology as shown in Figure 7(a), which depicts the solution computed for $H/T = 2.50$. The lateral banks determines an unavoidable wave reflection, which eventually leads to an increase of the ship resistance because of the augmentation of its wave component. This effect is even more severe in the shallower flow cases, when the wave magnitudes get higher, as depicted in Figs. 7(b) ... (d).

The reason behind this phenomenon resides in the pressure drop which occurs beneath the hull. For the smallest under-keel clearance and therefore the lowest water level in the channel, the free surface deformation effects are the largest, due to the largest blockage. Here the blockage indicates the amount of space a ship takes in the cross section of the fairway. Conventionally, the blockage is defined as the ratio between the cross sectional areas of the ship and the fairway. This is the ratio between the underwater part of the ship and the water vertical surface measured in a midship cross-section. From these figures, differences can be seen, showing especially a larger low-pressure area around the midship with negative consequences not only on the hydrodynamic resistance, but also on the wave system that propagates laterally and downstream.
Figure 7. Free-surface elevation computed for \( Fr = 0.195 \) for the restricted flow. (a): \( H/T = 2.50 \), (b): \( H/T = 2.0 \), (c): \( H/T = 1.50 \), (d): \( H/T = 1.25 \).

In the under-keel clearances of 25% and 50% of the draught, see Figs. 7 (c), (d), due to the blockage the flow is deflected around the hull and the pressure reduces significantly. Aside the risk of the breaking occurrence, another effect of the lower under-keel clearances is the generation of a prominent bow wave crest, which will contribute to the resistance increase because of the additional power consumption to be carried by the moving hull. Similar findings were reported in [6] and [14], although the hulls considered there were defined by higher block coefficients.

Since it has been stated above and in [14] that responsible for the change of the wave train is the pressure drop beneath the keel and around the ship hull caused by the limited navigation conditions, Figure 8 proposes a comparison of the relative pressure signatures on the bottom of the channel, computed for the same four water depths as in Figure 7. Obviously, the shallower the water get, the more prominent the loss in the computed relative pressure around the central part of the hull is found. The pressure drop induces not only the squat of the ship, which is in fact a sinkage increase, but also a modification of the trim angle, with a significant influence on the augmentation of the total ship resistance, as it will be show later in this paper. Summing up, the region of the low pressure around the hull may be directly linked to the wave troughs developed in the midship region, as shown in Figure 7.

Figure 9 shows the computed sinkage solutions drawn versus the water depth ratio \( H/T \) at the same smallest ship-to-bank distance \( B/L_{pp} = 0.2 \). In general, the sinkage increases with the decrease of water depth, and especially at a water depth less than \( H/T = 1.5 \), the sinkage changes more sharply revealing
a significant shallow water effect. The computations performed for the other ship-to-bank distances at the same small water depths revealed a similar tendency: when the hull moves closer to the bank, both sinkage and trim increase. The bank only affects the pressure distribution on the hull if the distance between ship and bank is sufficiently small, in this particular case, $B/L = 0.2$. Computations not reported in here have confirmed that if the ship-to-bank distance exceeds this value, no significant influences of the bank on the forces and moments on the ship hull could be observed.

![Figure 8](image_url)

**Figure 8.** Relative pressure distribution on the water bottom computed for $Fr=0.195$ for the restricted flow. (a) $H/T = 2.50$, (b) $H/T = 2.0$, (c) $H/T = 1.50$, (d) $H/T = 1.25$.

As mentioned above, the second significant effect of the shallow water navigation and the ship-to-bank proximity regards the augmentation of the hydrodynamic resistance, with direct effects on the fuel consumption, therefore on the shipping financial efficiency. In this sense, Figure 10 proposes a comparison of the computed resistances for six $Fr$ numbers in calm water, four shallow water depths and for a given width of the fairway. For the highest $Fr$ number of 0.205, which corresponds to a speed of 20.14 knot, the total resistance is more than 50% higher than one for the corresponding unrestricted water navigation case as the same Froude number. From this point of view, when navigating in channels, the ship speed has therefore to be reduced significantly for obvious economic reasons. In the particular case of the study reported herein, to keep the same hydrodynamic resistance, the ship speed has to be reduced to the outmost value of 17.9 knot, which corresponds to a $Fr$ number slightly larger than 0.18.

Figure 11 shows the wave cuts on the ship hull computed for five different $Fr$ numbers, where the hull extends from $x/L_{pp} = 0$ to $x/L_{pp} = 1$. Noteworthy to mention that because of the squat
phenomenon most of the waves developed on the hull are of a negative magnitude, except for the extremities where small crests are developed locally. Another key point that should be emphasized here is that regardless of the $Fr$ number value at which the computation is performed, the number of the crests is about the same and the wave geometries are distorted, being rather far from that of a periodic function which is usually expected to look like [16]. Obviously, this is due to the interference with the banks of the channel, which induces a significant wave interference, as mentioned above when Figure 7 was discussed.

From the environmental point of view, a major concern is related to the risks of the occurrence of the banks erosion produced by the waves generated the moving ship. The wave train energy is directly related to the wave amplitudes and phase speed. Because of the strong nonlinearities induced by the complex wave reflections and interferences, there are no analytic solutions based on which a prediction can be possible. Under such circumstances, the only available method is the numerical approach. Based on the numerical solutions computed for the six $Fr$ numbers, in the followings the wave profiles for the lowest ship speed ($Fr = 0.155$) and for the maximum possible one ($Fr = 0.185$) are analyzed in Figs. 12 and 13.
Figure 12 shows the wave profiles computed at the downstream of the ship, drawn in the centerline of the channel, whereas in Figure 13 they are drawn on the side bank of the fairway. In terms of the wave system developed in the symmetry plane of the computational domain shown in Fig 12 it may be noticed that the highest wave amplitude corresponds to a region that extends from $x/L_{pp} = 1$ up to $x/L_{pp} = 1.8$. The non-dimensional highest crest is slightly larger than $0.0095 L_{pp}$, which means about 2.185 m at full scale for the ship speed corresponding to $Fr = 0.185$. Needless to say that, even though the wave is computed in the longitudinal axis of the computational domain, this value is regarded as being rather high for a restricted water navigation case. Figure 13 reveals that for both $Fr$ numbers flow cases considered, at the downstream of $x/L_{pp} = 1.75$, a water level increase is observed, a fact which can be linked to the piston-like effect. This is resulting in a longer wave shape because the water level increases while the ship model is travelling through the canal section and at the same time pushing water out of the canal considered section.

![Figure 12. Wave profiles computed for $Fr=0.185$, $H/T=1.25$, $B/L_{pp}=0.2$. in the centerline of the free-surface.](image1)

![Figure 13. Wave profiles computed for $Fr=0.185$, $H/T=1.25$, $B/L_{pp}=0.2$. on the side bank of the channel.](image2)

If the water level would have been computed for a longer period of time (after the ship model passed through the considered section) then a water level drop would occurs because of the viscous effects. Waves of about $(0.004 - 0.005)L_{pp}$ in magnitudes, which means about 1.04–1.3 m at full scale are seemingly to develop if the ship move at $Fr=0.185$. Since such waves, whose amplitude are far too high from an acceptable level, may have a very aggressive actions on the banks, the recommended speed the ship moves at, has to be limited at values well below that limit, i.e. at a level that should not exceed the maximum affordable limit that the banks may support the generated wave train. In all the restricted navigation cases such a conclusion eventually leads to a set of prescriptions for regulating the traffic conditions such that the influence on the environment has a minimum possible effect. Obviously, such regulatory restrictions regards the maximum allowable speed the ship may move at in that particular channel.

5. Concluding remarks
The reported study proves that the flow pattern around a ship hull moving in restricted conditions (laterally as well as in terms of the water depth) is considerably affected not only by the water depth, but also by the proximity of the lateral banks of the channel. In open and unrestricted water the flow is fully three dimensional, as a significant part of the displaced water flows under the unlimited domain below the hull. Nonetheless, the computation performed within the present study proved that the stream pattern around the considered ship hull sailing in a shallow water tends to become mostly a two dimensional one.
As a general conclusion, when a ship is sailing along the midline of a symmetric channel it will encounter an increased resistance and a modified sinkage and trim as a result of the increased relative velocity between the water and the ship due to the increased blockage, but no lateral forces will be induced by the banks because of reasons of symmetry. Banks are only affecting the pressure distribution on the ship hull if the distances between ship and bank are sufficiently small.

Summing up, based on the discussions provided in the above sections of the paper, the following conclusions may be put forward:

- The grid convergence test proved that for obtaining sufficiently low discretization errors, grid refinement is recommended;
- Wave heights are strongly affected by depth to draft ratio;
- Computed values of the ship squat were in the range between 0.009 and 0.019 m at the model scale for the maximum ship speed in shallow water;
- The viscous resistance increases strongly in shallow water, in connection with an increased over-speed along the hull, a more horizontal flow along the hull and larger pressure gradients, and ultimately the occurrence of flow separation;
- The total hydrodynamic resistance increases significantly in confined water because of the larger pressure gradients, and of the occurrence of flow separation;
- The proximity of the solid walls requires a drastic limitation of the ship speed in order to avoid the generation of high waves that may eventually affect the integrity of the channel banks.

6. Acknowledgements
The computations were performed on the HPC at the “Dunarea de Jos” University of Galati. A special thank goes to A. Istrate for the unfailing support in providing optimal access to the computational resources when still available to the scientific community.

7. References
[1] Raven., H. C., 2012, 29th Symp. on Naval Hydrodynamics, Gothenburg, Sweden
[2] Chen, H. C., Liut, D. A., Hwang W.Y. and Lin, W. M., 2003, Int’l. Conf. on Marine Simulation and Ship Manoeuvring (MARSIM), Japan
[3] Lo, D. C. Su, D.T. Lin, I. F., 2009, 19th Int’l Offshore and Polar Eng. Conf. Osaka, Japan
[4] Zou, L. Larsson, L., Delefortrie, G. and Lataire, E., 2011, 2nd Int’l Conf. on Ship Manoeuvring in Shallow and Confined Water: Ship to Ship Interaction. Trondheim, Norway
[5] Zou, L. and Larsson, L., 2013, Int. Shipbuild Prog., 60 pp. 1–4
[6] Van Hoydonck, W., Toxopeus, S., Eloot, K. Bhawsinka, K., Queutey, P. and Visonneau, M., 2019, J. of Marine Sci. and Technology 24
[7] Lungu, A., 2020, Procedia Manufacturing, 46, 432–439, doi: 10.1016/j.promfg.2020.03.063
[8] Larsson, L., Stern, F., Visonneau, M., Hirata, N., Hino, T. and Kim J., 2015, Tokyo 2015 Workshop on CFD in Ship Hydrodynamics, http://www.t2015. nmri.go.jp.
[9] Lungu, A., 2018, Proc. of the ASME 37th Int. Conf. on Ocean, Offshore and Arctic Engineering (OMAE), paper OMAE2018-77767, doi:10.1115/OMAE2018-77767
[10] Lungu, A., 2020, Journal Offshore Mech. Arct. Eng., 142(2): 021905
[11] Lungu, A. and Pacurar, F., 2009, AIP Conference Proceedings 1168, 693, https://doi.org/10.1063/1.3241559
[12] Lungu, A., Raad, P. E. and Mori, K., 1997, Proc. of the ASME FED Summer Meeting, ASME FEDSM97-3404
[13] Wackers, J., Deng, G. D., Leroyer, A., Queutey, P. and Visonneau, M., 2014, Comput. Fluids 92
[14] Lungu, A., 2007, AIP Conference Proceedings 936, 647, https://doi.org/10.1063/1.2790231
[15] Pacurar, F., Lungu, A. and Marcu, O., 2011, AIP Conference Proceedings 1389, 191, https://doi.org/10.1063/1.3636699