Numerical results of CFD simulations for full-scale ship-shape body motion in deep and shallow water

Pavel Ivanov
Nikola Vaptsarov Naval Academy, Varna, 9026 Bulgaria
p.ivanov@nvna.eu

***Abstract**: The objective of the simulations is to calculate the total resistance and visualize the flow around a bare hull of a boat motion in deep and shallow water. Ship resistance is an important issue for ensuring smooth propulsion in various speeds in open sea and especially in the increasing influence of shallowness. Subcritical speed range in open shallow water is the main interest regime. A limited number of cases in critical and supercritical speed regions are applied in computations for comparison. For the full-scale bare ship-shape body resistance tests, computational fluid dynamics techniques and SST k-omega turbulence viscous model were employed. All calculations were carried out considering the influence of the free surface effect in a fluid. For convenience of mesh generation, an automated structured polyhedral meshing was adopted for the delicate curved surface and the domain. CFD solver interFoam – OpenFOAM, is applied to execute all computations in this article. The numerical model was initially built in FreeCAD, an open computer aided design (CAD) software and discretized in the cfMesh, a library implemented on top of the OpenFOAM framework. Post-processing is performed within ParaView, an application for scientific visualization. All software is free.*

***Keywords**: SIMULATION, FREE SOFTWARE, CFD, FREECAD, OPENFOAM, PARAVIEW*

1. Introduction

Maneuverability is a key component in performance. There are standards for ship maneuverability that are promulgated by the International Maritime Organization (IMO), but these standards cannot ensure absolute navigation safety. For this purpose, ITTC Maneuvering Technical Committee take responsibility for the prediction of the hydrodynamic performance of ships and marine installations based on the results of physical and numerical experiments. The test process is complex and the distribution information of the flow field around the ship obtained from the test is limited. Compared with traditional methods, computational fluid dynamics (CFD) method can provide more detailed information for naval architecture and ocean engineering, Yang et al [1].

CFD simulations not only minimize physical testing which is expensive but also reduce time spent for field tests. More importantly it is achievable to test the models in environments and situations that are hard to experiment within real life. Simulations can recreate critical moments or incidents cases virtually and this may increase understanding and decrease the chance of such.

Obtaining data from CFD models for shallow waters, gathering information and visualizations about hull behavior can be helpful for seamen.

The behavior of the ship in real sailing conditions is important for safety, where the total resistance gives us the proper speed to optimize the voyage. Therefore, use of numerical methods to predict the hydrodynamic phenomena during maneuverability in shallow waters can help navigators to optimize their decisions, consent within review'24 [2].

It is noticeable that there is a relatively small amount of research papers that considered the free surface effect of flow around the vessel, which is closely related to resistance and maneuverability. In the present study, the flow field around ship-shaped body is simulated.

2. General Setup

A hydrodynamic coefficient must be determined to predict the maneuvering motion of the ship. The methods used for this purpose are the captive model test, system identification techniques applied to the results of free-running model tests, and computational fluid dynamics (CFD).

CFD simulations became popular with the advancement of computer technology. The results obtained with the CFD method are very reliable and provide a detailed picture of surface elevations as well as velocity and pressure fields. This lead to a better understanding of the hydrodynamic phenomena of a ship maneuvering in shallow waters. CFD calculations are an alternative to physical methods because they provide reliable data at a much lower cost [3].

In the basis of all models is momentum, mass, and energy balance which are PDEs (Partial Differential Equations). They require Boundary and Initial conditions to be defined in the solver before calculations.

The final system of differential, integral and algebraic equations is complex to be solved analytically and for this reason, numerical solutions are applied based on appropriate algorithms, where the initial mathematical model is reduced to a system of linear algebraic equations giving an approximate solution.

Finite Volume Method (FVM) is the common approach that is applied for discretization of the space under study. OpenFOAM implements a cell-centered FVM.

2.1 Case formulation

The precise formulation is a fluid mechanics of a ship-shape body with surge motion in deep and shallow waters, representing a straight flow around a static body (captive model).

The test model simulates forward motion with a speed U = 2, 3.2, 4 and 5.2 m/s with drift angles of 0°, 3°, 6°, 9° and 12°in calm water.

The Froude number, for hull resistance indication (F_n) and for shallow waters (F_{n_h}):

$$F_n = \frac{U}{\sqrt{gL}}$$

where U is the ship velocity, L is the ship length, g is acceleration due to gravity. A general rule is that Fn < 0.4 indicate a pre-planing hull, also known as displacement hull, and Fn > 1.0 indicate a fully planing hull.

$$F_{n_h} = \frac{U}{\sqrt{gh}}$$

where h is depth of water basin. Setting relative depths for h/d to 4.0 (deep water), 1.8 (moderate shallow water), 1.5 (shallow water) and 1.2 (marginal shallow water) are specified by Vantorre et al [4].

For $F_{n_h} < 1$ the flow is called a subcritical flow, further for $F_{n_h} > 1$ the flow is characterized as supercritical flow. When $F_{n_h} \approx 1$ the flow is denoted as critical flow.

The numerical study considers F_n of 0.10, 0.16, 0.20, 0.26 for speed in deep water and F_{n_h} from 0.17 to 0.84 for shallow water.

2.2 Mathematical models

The mathematical model of ship maneuvering motions is a key component in marine simulators. The numerical results presented in this paper are used to improve understanding of hydrodynamic phenomena acting upon hull in shallow waters with seawater free surface effect.

Therefore:

- RANS method is used to predict the hydrodynamic forces and moments to obtain a more accurate fluid flow equation since it capable to capture turbulent effects [5].
- Turbulence model 'SST k-Omega' is used in OpenFOAM [6]. The Menter [7] Shear Stress Transport model (SST) k-ω turbulence model which the transport equations are solved for two variables: turbulent kinetic energy k and specific dissipation rate ω.

The maneuvering derivatives of CFD data from such settings help to improve the accuracy of mathematical models for ships maneuvering motions and enable maneuvering motions to be predicted, confirmed by Yang et al [1].

❖ ***General assumptions:***

Mathematical models describing ship manoeuvrability and seakeeping are based on six equations of motion according to the six degrees of freedom (6DOF) of a ship and an equation of equilibrium for ship propulsion.

A model motivated by Newton's second law: F = Δa, where F represents force, Δ the mass of the vessel and a - the acceleration of ship dynamics, can be described by means of strongly nonlinear dimensional equations from figure 1.

$$\Delta(\dot{u} + qw - rv) = X$$

$$\Delta(\dot{v} + ru - pw) = Y$$

$$I_z\dot{r} + (I_y - I_x)pq = N$$

where Δ - mass of the ship, u, v, w – linear velocity and angular velocity p, q, r. Moments of inertia *Ix*, *Iy*, *Iz* around X-Y-Z axis which are ship-fixed reference frame and Xo-Yo-Zo are the Earth-fixed reference frame. Note here that *X and Y* are total forces acting along appropriate axes and *N* is total moments acting to the hull.

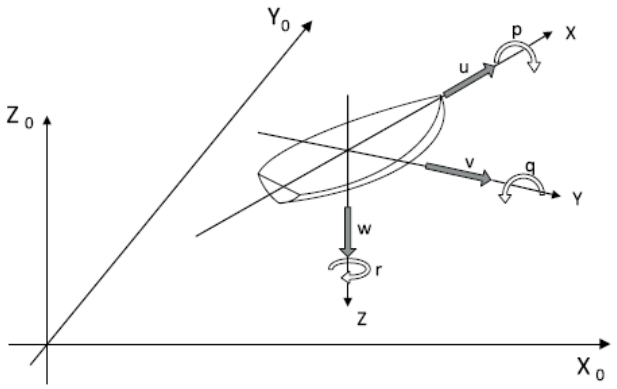


Fig.1 *6DOF for a ship-shape body with centre of rotation at the ship gravity center exactly in the mid-section.*

For many control applications like the heading, speed and/or trajectory stabilization it is a common practice to neglect pitch, roll and heave motion with assuming that $p = q = w = 0$. That lead to change of the 6 DOF model to the 3 DOF model:

$$\Delta(\dot{u} - rv) = X$$
$$\Delta(\dot{v} + ru) = Y$$
$$I_z\dot{r} = N$$

❖ ***Formal models:***

The equations of motion in the horizontal plane, introduced by Abkowitz in the 1960s [8], are based on a third order Taylor-series development of the hydrodynamic forces. Unfortunately, a 3rd order Taylor-series expansion results in a large number of terms. By applying some physical insight, the complexity of these expressions can be reduced. Abkowitz (1964) makes the following assumptions:

1. *Most ship maneuvers can be described by a 3rd-order truncated Taylor expansion about the steady state condition* $u = u_0$
2. *Only 1st-order acceleration terms are considered.*
3. *Standard port/starboard symmetry simplifications except terms describing the constant force and moment arising from single-screw propellers.*
4. *The coupling between the acceleration and velocity terms is negligible.*

Simulations of standard ship maneuvers show that these assumptions are quite good, Fossen (2011) [9]. The force of hydrodynamic phenomena related to the hull movement through the water can be expressed as follow:

$$X = X_0 + X_{vv}v^2 + X_{vr}vr + X_{rr}r^2$$

$$Y = Y_v v + Y_r r + Y_{vvv}v^3 + Y_{vvr}v^2 + Y_{vrr}vr^2 + Y_{rrr}r^3$$

$$N = N_v v + N_r r + N_{vvv}v^3 + N_{vvr}v^2 + N_{vrr}vr^2 + N_{rrr}r^3$$

here X_0 is direct navigation resistance and we can *note also that equation *N* is as same as *Y*.

During direct navigation, the ship-shape body moves at constant speed along the X direction, and the hull resistance coefficient X_0 is measured.

When the ship moves obliquely at a certain drift angle β, the velocity components u and v in the body-fixed coordinate system can be expressed as:

$$u = U\cos\beta$$
$$v = -U\sin\beta$$

where hull has longitudinal velocity u and transverse velocity v, and the other disturbances ($\dot{v} = \dot{r} = r = 0$) are zero, than Taylor expansion can be simplified (regressed) as follows:

$$X = X_0 + X_{vv}v^2$$

$$Y = Y_v v + Y_{vvv}v^3$$

Here X_{vv}, Y_v, Y_{vvv}, N_v and N_{vvv} are transverse velocity dependent damping coefficients (*equation for *Y* is same as *N*). The coefficients Y_v and N_v are the linear coefficients and the rest are nonlinear ones. Simulation at various drift angle β provides the forces *X*, *Y* and *N* moment (Fig.2).

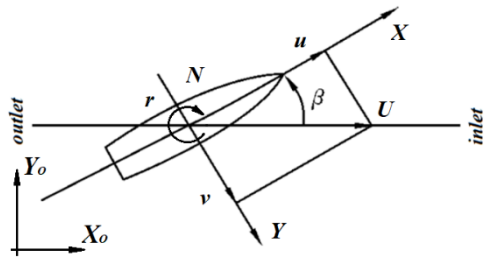


Fig.2 3DOF Diagram of oblique towing test.

Considering the port- and starboard symmetry, the mathematical model relating *Y*, *N*, and *v* can be expressed as:

$$Y = Y_0 + Y_v v + \frac{1}{6}Y_{vvv}v^3$$

$$N = N_0 + N_v v + \frac{1}{2}N_{vvv}v^3$$

where Y_v, Y_{vvv}, N_v, N_{vvv} are the maneuvering hydrodynamic derivatives, and Y_0, N_0 are the lateral force and moment at the initial steady state of forward motion.

The variables are transferred to non-dimensional values based on the ship length *L*, ship speed *U*, and water density ρ. They are distinguished by putting the sign of $'$ (prime) as follows:

$$Y' = Y'_0 + Y'_v v' + \frac{1}{6} Y'_{vvv} v'^3$$

$$N' = N'_0 + N'_v v' + \frac{1}{2} N'_{vvv} v'^3$$

The dimensionless forms for wetted surface:

$$v' = \frac{v}{U} \quad F' = \frac{F}{0.5\rho L d\, U^2} \quad M' = \frac{M}{0.5\rho L^2 d\, U^2}$$

❖ ***Modular mathematical models:***

In the 1980s the 'concept of modularity based on a separate representation of elements of the manoeuvring model' has been fully recognized among the world, MARSIM '93 [10]. The principal structural form of a modular manoeuvring model neglecting the external forces is based on Newton's equations:

$$\Delta(\dot{u} - vr - x_G r^2) = X$$
$$\Delta(\dot{v} + ur + x_G \dot{r}) = Y$$
$$I_{zz}\dot{r} + \Delta x_G(\dot{v} + ur) = N$$

There are a lot of expressions of this mathematical model comment a research group called Maneuvering Modeling Group (MMG) in Japan [11]. Nowadays, a lot of simulation methods based on MMG model exist and many simulation methods sometimes harm the adaptability of hydrodynamic force data for the maneuvering simulations since one method may be not applicable to other methods in general.

To avoid this, the basic part of the method should be common. The test procedure and the data analysis to determine the hydrodynamic force coefficients for the simulations should be also common since those often involve the quantitative value of the hydrodynamic coefficients (*next equation Δ sign will substitute mass symbol *m* used in MMG model for synchronous purposes and other reasons):

$$(\Delta + \Delta_x)\dot{u} - (\Delta + \Delta_y)v_m r - x_G \Delta r^2 = X$$
$$(\Delta + \Delta_y)\dot{v}_m + (\Delta + \Delta_x)ur + x_G \Delta \dot{r} = Y$$
$$(I_{zG} + x_G^2 \Delta + J_z)\dot{r} + x_G \Delta(\dot{v}_m + ur) = N_m$$

where Δ_x, Δ_y is added mass, v_m is lateral velocity at midship. x_G longitudinal coordinate of center of gravity of ship, I_{zG} is moment of inertia of ship around center of gravity and J_z added moment of inertia.

❖ ***Mathematical model describing manoeuvring motion:***

The research work published by authors during the last and this century is of great value for the prediction of ship manoeuvrability. However, none of the existing methods is sufficiently broad and accurate at present to be raised up to a general, standardized manoeuvring model, Kose (1995) [12].

The preference is given to a modular manoeuvring model. In contrast with the formal or regression models, ship/fluid interaction is no longer treated as a black box and the hydrodynamic forces are linked to the three modules of a manoeuvring ship: hull, propeller and rudder, Eloot (2006) [13].

3. CFD Simulation

The physical model of environment domain was created in FreeCAD version 0.21.1. Analyses were performed using the software product OpenFOAM version 2212. Post-processing and images are extracted from ParaView version 5.12.0.

3.1 Open∇FOAM Solver

The multiphase solver ***interFoam*** is designed for transient simulations for two incompressible, isothermal, and immiscible fluids. It can handle laminar and turbulent, accommodating both Newtonian and non-Newtonian fluids. It utilizes the Volume of Fluid (VOF) approach for capturing the interface between the fluids accurately.

The solver uses the PIMPLE (merged PISO and SIMPLE) algorithm for pressure-momentum coupling. This algorithm leverages the strengths of both PISO and SIMPLE methods for pressure-velocity coupling, ensuring robustness in handling transient flows with large time steps. This approach is supplemented by under-relaxation techniques to secure convergence stability.

interFoam uses the multidimensional universal limiter for explicit solution (MULES) method to maintain boundedness of the phase fraction independent of underlying numerical scheme, mesh structure [14]. So, the choice of schemes for convection are not restricted to those that are strongly stable or bound.

The ability to capture complex fluid interactions together with dynamic mesh capabilities makes it very versatile and useful in the marine industry, where complex ship-waves interactions can be studied. For the transportation industry, mixing stirring tanks can be analyzed and tank sloshing phenomena. Wave loading or offshore structure loads can be predicted.

As a validation of *interFoam* solver, it is proposed to recreate a Sloshing Problem. As an example, a 2D and 3D Dam Break problem are applied.

3.2 Sloshing Problem: Dam break problem

The 2D dam break case is represented by OpenFOAM tutorials in software package after installation the product or from official website [15]. A coarse mesh with the same domain dimensions was created for comparison, figure 3a.

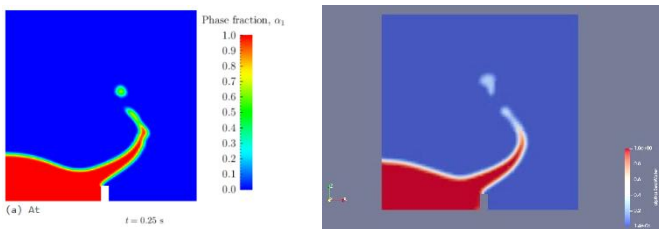


Fig.3a *2D breaking of a dam compared with experiment (time=0.25 sec)*

where the sample is taken from *OpenFOAM Tutorial number 4.1.*

The 3D dambreaking problem developed by Kleefsman et al [16]. A coarse mesh with the same domain dimensions was created for comparison, figure 3b.

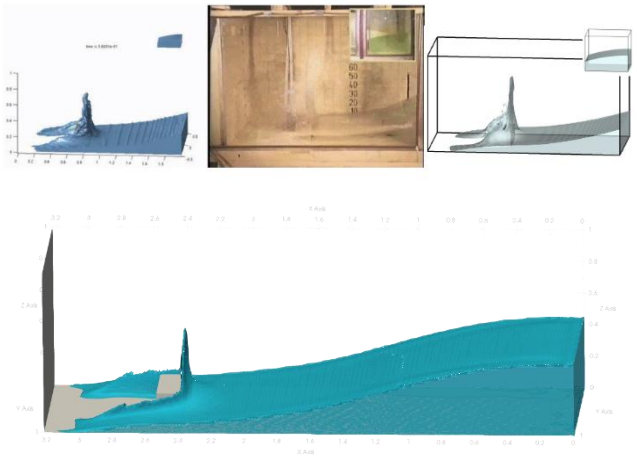


Fig.3b *3D coarse mesh snapshots of a dambreak simulation with a box in the flow compared with experiment (t=0.56 sec)*

It is visible and we can say for the used approach of CFD method is correct and reproduces the test case very well, which is a guarantee for the authenticity (verification) of further research on the ship-shape body.

4. Ship-shape body and Simulation design

4.1 Physical shape of the studied body

The physical model used in the simulation is in figure 4.

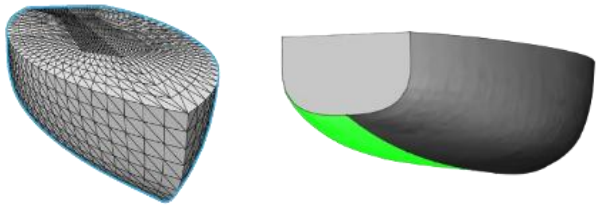


Fig. 4 *Body shape evolution*

All simulations are performed with the bare boat design model with following dimensions (Tab. 1)

Table 1. *Boat model particulars, Full-scale*

	Design	
Length of waterline	L_{WL}	41.21 m
Breadth	B	15.74 m
Draft	d	3.27 m
Wetted Surface Area	S_0	577.19 m^2

The scale is one to one, full scale because of viscous forces (ITTC, 1978) [17].

4.2 Domain numerical set-up

A 3D rectangular computation domain representing a numerical towing tank is used in the simulations. Due to the vertical plane symmetry of the ship hull and domain, the task is symmetrical so to facilitate memory and computation time, the simulation can be done for half of the experiment. The half domain will have length of 255 m, close to 2L in front and 3L behind the hull, 156 m wide and 26 m above water line. Depending on the case, there is a free surface at distance of 3.924 m from domain bottom for marginal shallow water (Fig. 5a).

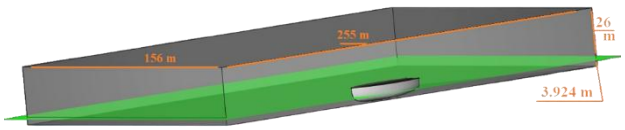


Fig.5a *View of half-model*

Both domains' dimensions are selected sufficiently large to avoid back flow at high drift angles. For full model distance of the inlet and outlet boundary from ship center is considered to be more than 2L and 4L, respectively. The side boundaries are located at around 3.5L and the top is at 1L from the free surface (Fig. 5b) while bottom distance depends on cases.

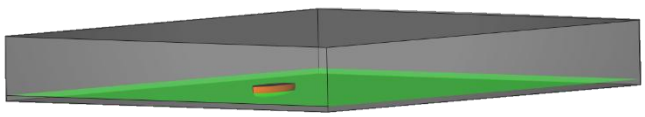


Fig.5b *The domain*

Full model dimensions are tailored considering ITTC recommendations (75-03-04-01: Guideline on Use of RANS Tools for Manoeuvring Prediction, 2017).

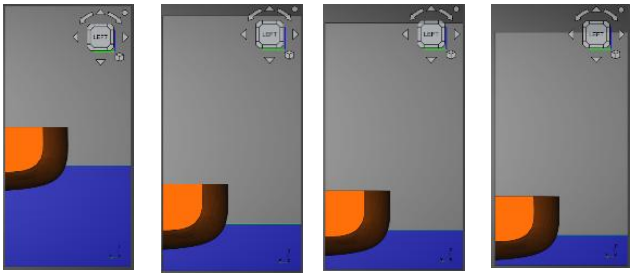
The model is prepared for calm water surface, no wind effects. It is achievable by dividing the inlet wall of physical domain to two zones, one for water inlet and the other zone for air inlet, which condition is added to top boundary. BCs are on table 2.

Table 2. *OpenFOAM built-in boundary conditions*

Boundary	alpha.SeaWater	p_rgh	U
inlet	fixedValue	fixedFlux Pressure	fixedValue
outlet	variableHeight FlowRate	zero Gradient	outletPhase MeanVelocity
top	inletOutlet	total Pressure	pressureInlet OutletVelocity
side	symmetry	symmetry	symmetry
bottom	zeroGradient	fixedFlux Pressure	movingWall Velocity
hull	zeroGradient	fixedFlux Pressure	fixedValue

The hull is a *no-slip wall* allowing for boundary layer around the ship-shape body to be developed. So y^+ wall treatment with 10 layers is used for the hull with expansion ratio 1.2 (ITTC 2017, Recommended Procedures and Guidelines). Bottom and side boundaries are modelled as *symmetry* for deep water case, while top is *open* boundary. As for shallow water only, the bottom is changed to *a no-slip wall.*

Different model cases of depth can be seen in figure 6.



h_1=13.080m h_2=5.886m h_3=4.905m h_4=3.924m

Fig. 6 *Four depths: from deep to optimal shallow water models*

4.3 Mesh generation

Mesh is of highest importance in CFD simulations and the process of creating optimal mesh for any case is a hard and time-consuming process. Having this in mind, the quality metrics must be checked with an OK mesh Check Test for topological errors or orthogonality, skewness, aspect ratio and smoothness for obtaining accurate results.

Table 3. *Grid Quality Check for $h = 1.2d$, drift 12°*

Mesh: OK	Values:
Cells	6 164 509
Max Aspect Ratio	41.897
Max non-orthogonality [deg]	81.304
* faces > 70 [deg]	5 859
Avg non-orthogonality [deg]	8.058
Max Cell Skewness	3.798

Should be aware that numerical results can be considered precise and valid if its solution is independent of the grid. Nevertheless, validation and verification are mandatory.

5. Numerical results

- *Resistance test*

Drag coefficient, C_T can be calculated by the formula:

$$C_T = \frac{F_x}{0.5\rho S_0 U^2}$$

at different water depths, which gradually increases as the ship manoeuvring in shallower basin due to the effect of the sea bottom.

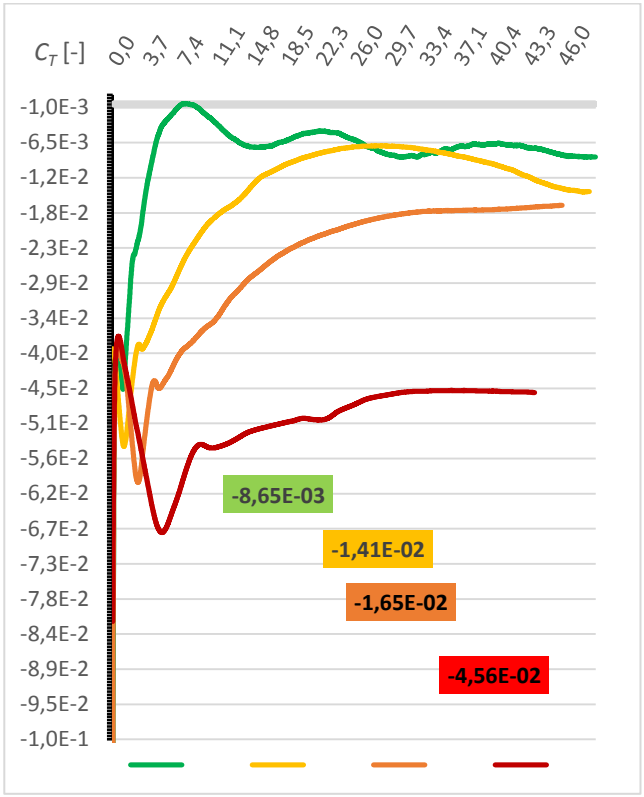


Fig.7 *Force coefficient* C_T *against time, within four depths at same ship speed U=5.2 m/s.*

Only for deep water with different speeds is drawn at figure 8:

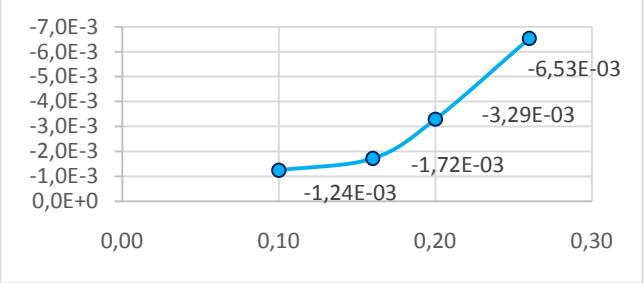


Fig.8 Results of resistance at various ship speeds in deep water.

here horizontally is Frude numbers (*Fn*) and vertically are the values of ship resistance coefficient in straight motion (C_T), which in FreeCAD coefficients.dat reporting function file correspond to *Cd*.

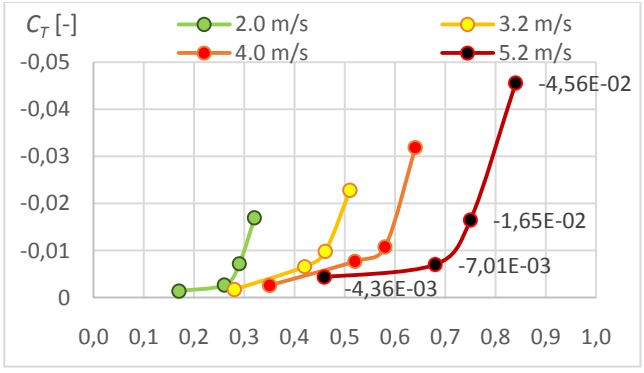


Fig.9a Results of resistance F_{n_h} in shallow water for dif. speed.

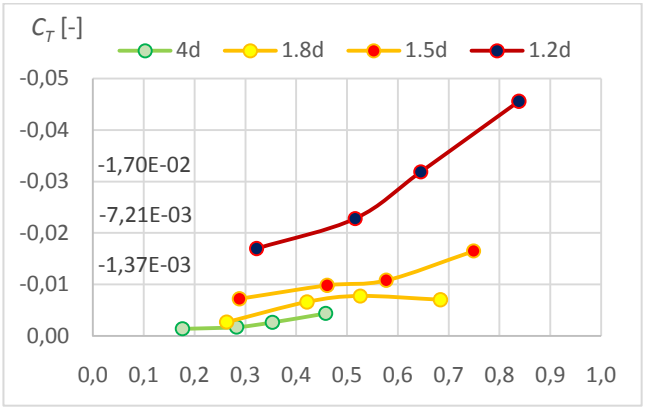


Fig.9b Results of resistance F_{n_h} at dif. speed for shallow waters.

Figure 10, gives us information about the values of force and moment for the whole period simulation time:

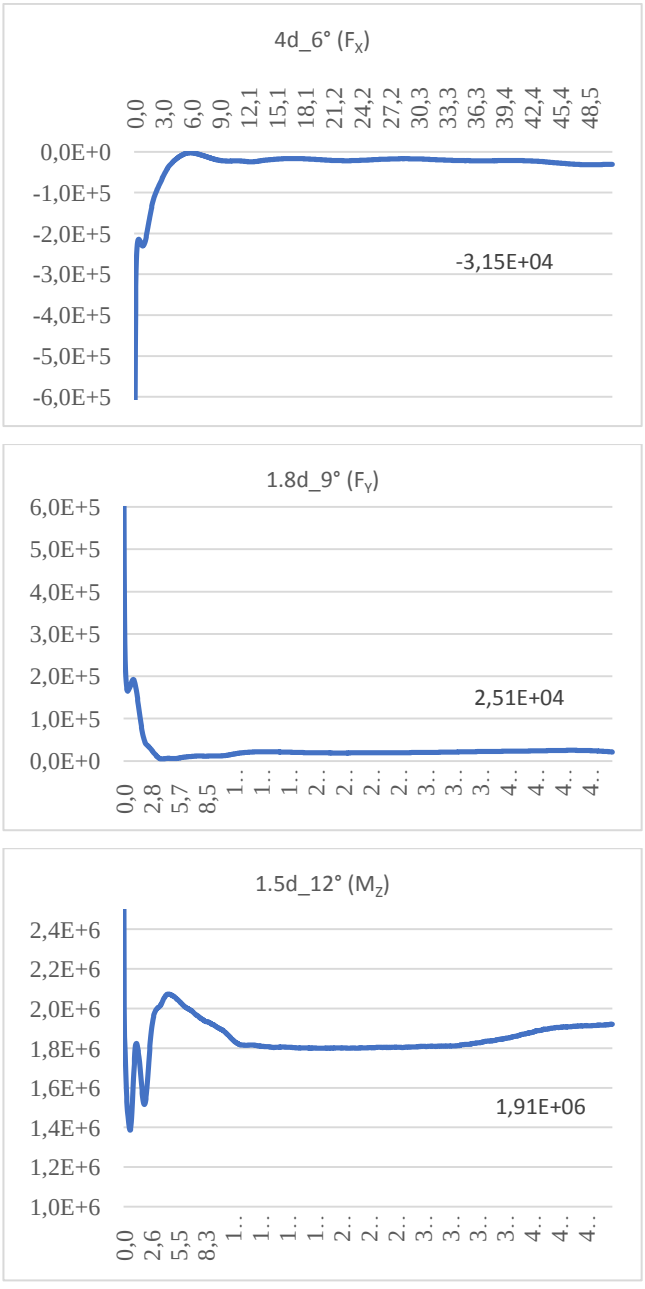


Fig.10 Report of functions (t=51 sec.)

- *Static drift test*

In the static drift test, the ship moves in oblique flow due to a given drift angle β. The hydrodynamic forces are a function of sway velocity, $v = -U\sin\beta$. Figure 11 illustrates the results of the static drift test at various water depths.

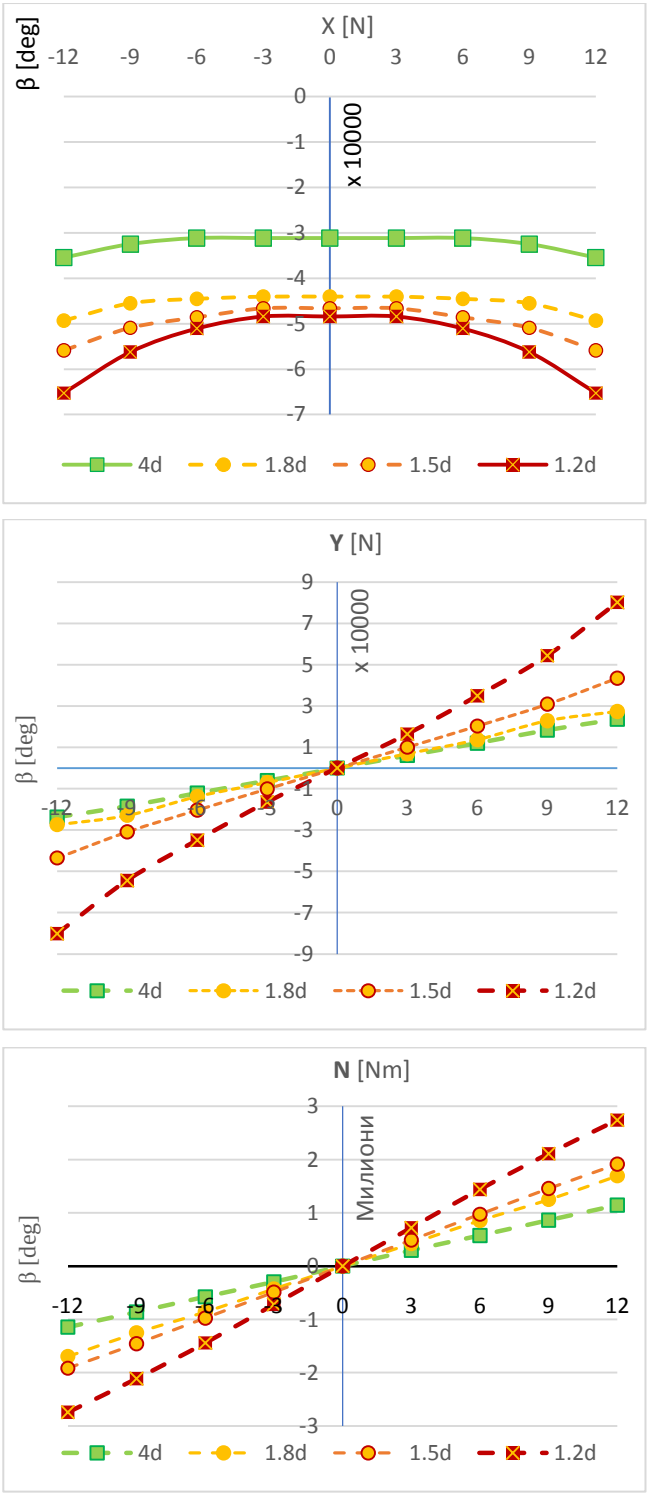


Fig.11 Results of static drift at various water depths: Surge force, Sway force, and Yaw moment. Hydrodynamic forces acting on hull.

The hydrodynamic forces are higher when the water shallowness increases. Diagram slope has a significant change at h=1.2d, as the hydrodynamic force are equal to double (2 times) of h=1.5d. The increase in hydrodynamic forces and moments in shallow water was caused by the increased in flow velocity and decrease in pressure through the limited space between ship bottom and seabed. This proves that shallow water affects considerably hydrodynamic forces generated by sway velocity.

5.1 Numerical post-processing (data visualization)

Free surface related patterns performance for different depths are shown on these figures, except figure 13 and 16.

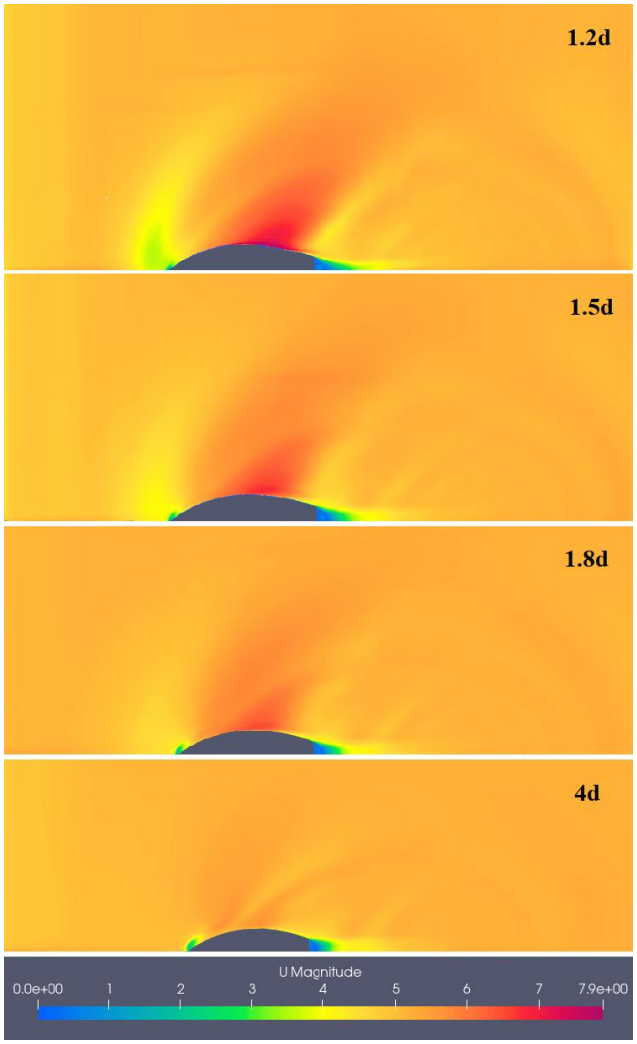
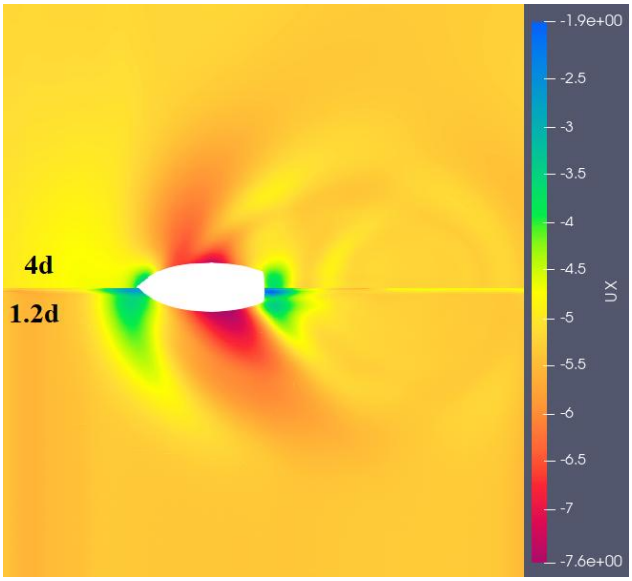


Fig.12 *Velocity field on interface-free surface*

Images of keel plane with top view are presented in figure 13.



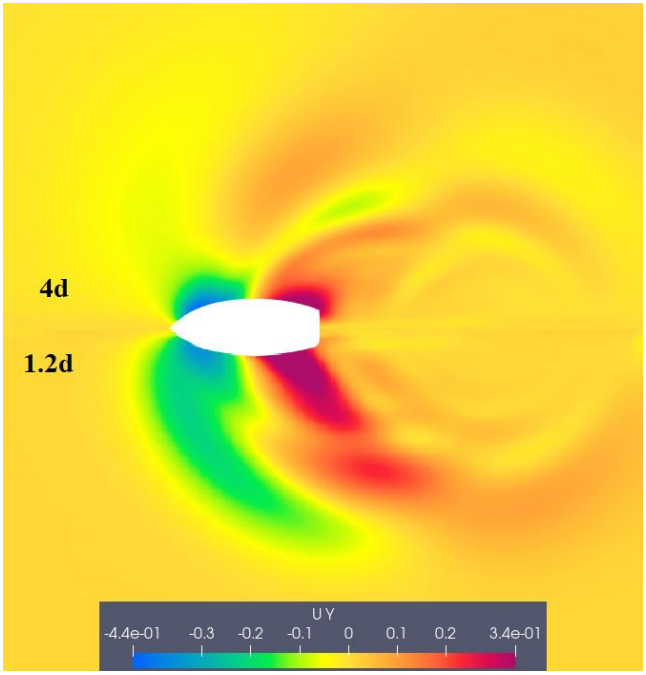


Fig.13 *Keel plane, velocity field at* X *and* Y *direction.*

here 'keel plane' means, that in ParaView a flat plane to Z axis is placed at the lowest construction point of ship hull.

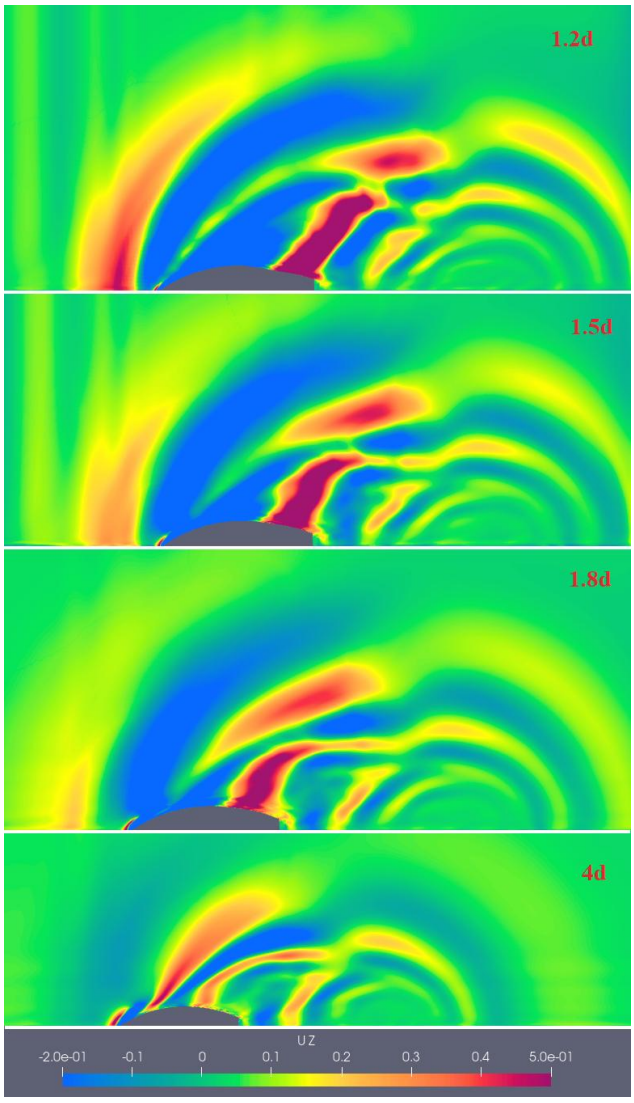


Fig.14 *Wave pattern, free surface U field at Z axis.*

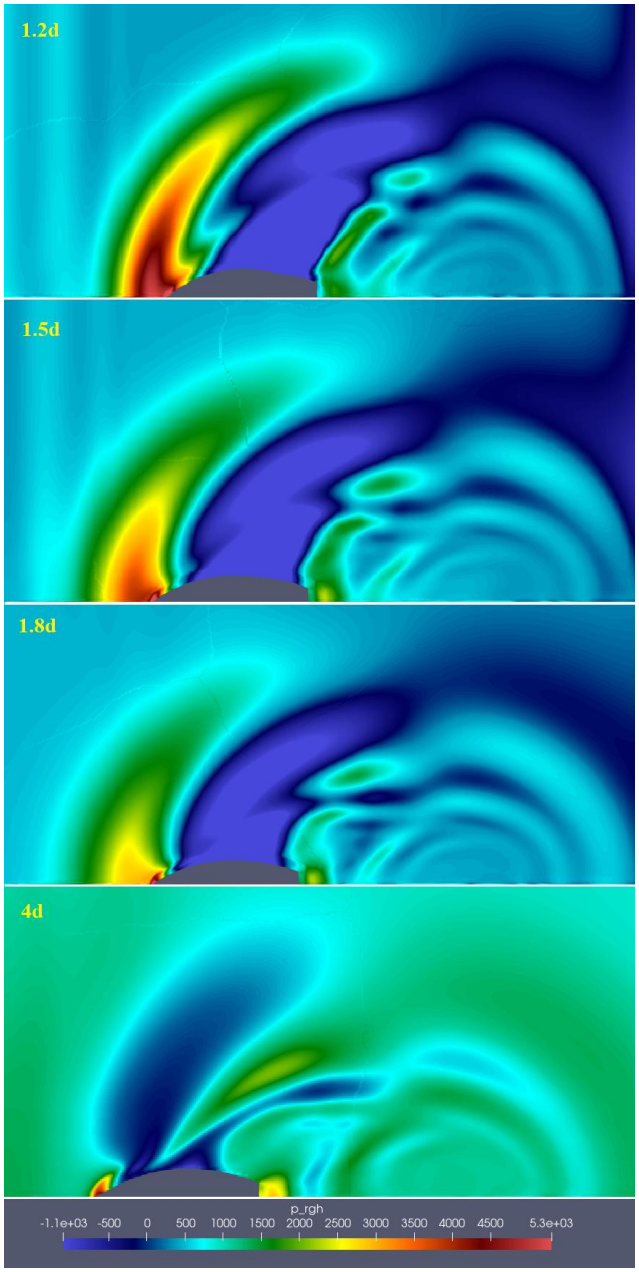
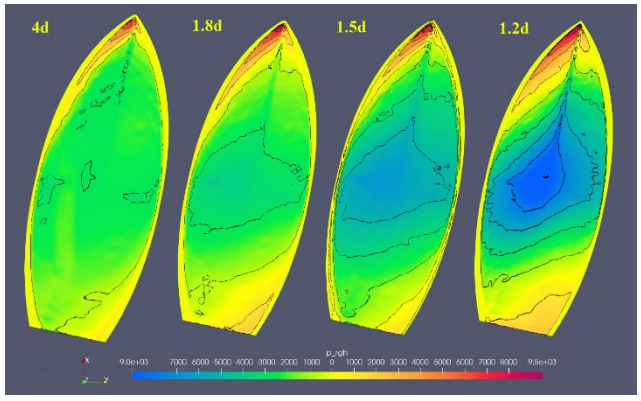


Fig.15 *Pressure field at free surface*

Images of pressure field at hull wetted surface on figure 13:



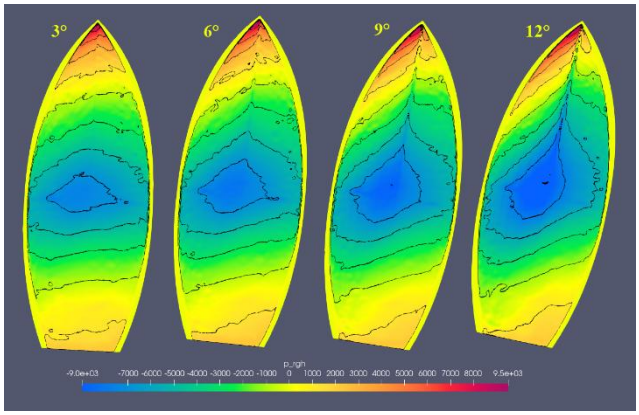


Fig.16 *Pressure field on hull bottom wetted surface at drifts with U=4m/s*

Applying vectors on free surface show velocity of the free surface interface, figure 17.

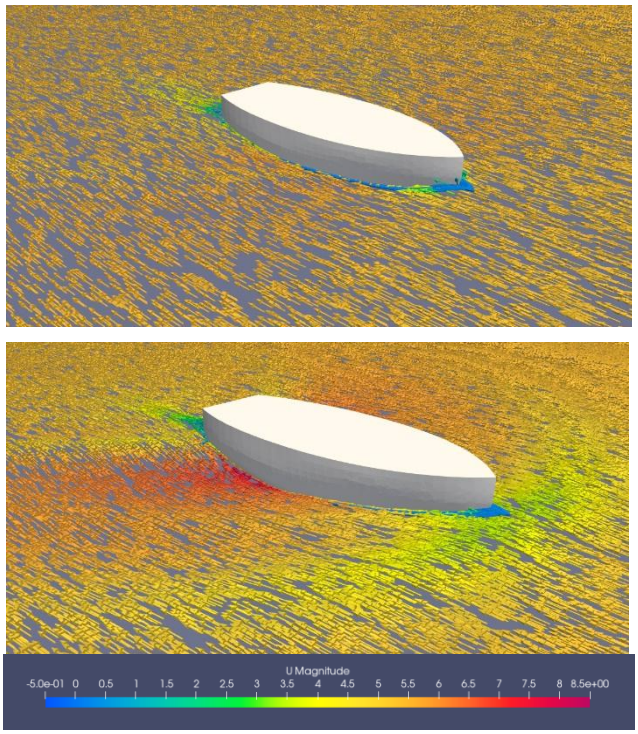
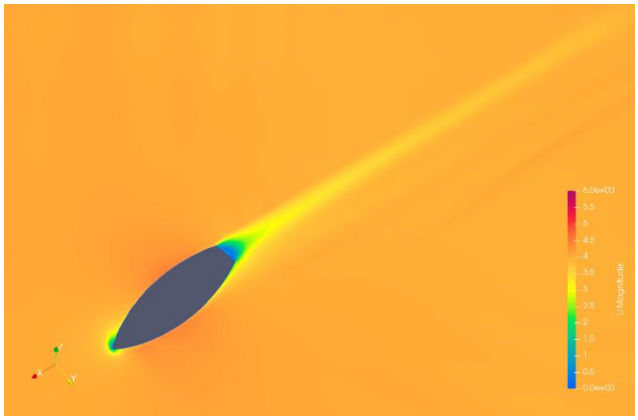


Fig.17 *Comparing deep water and marginal shallow water depths at 0 deg drift. U vector at free surface. U=5.2m/s.*



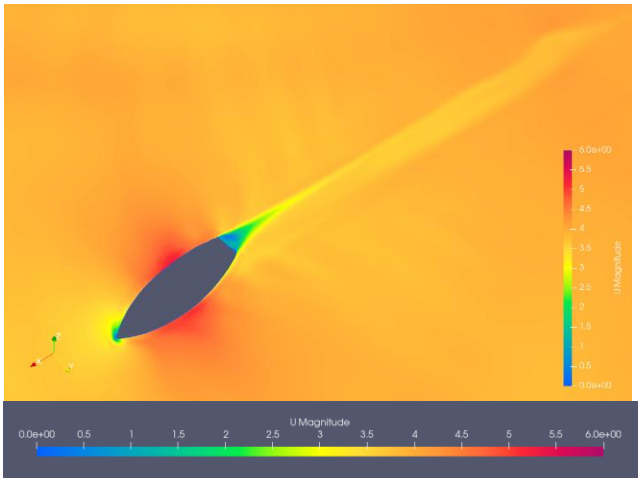


Fig.18 *Drift angle of 12 deg at deep and marginal shallow water.*

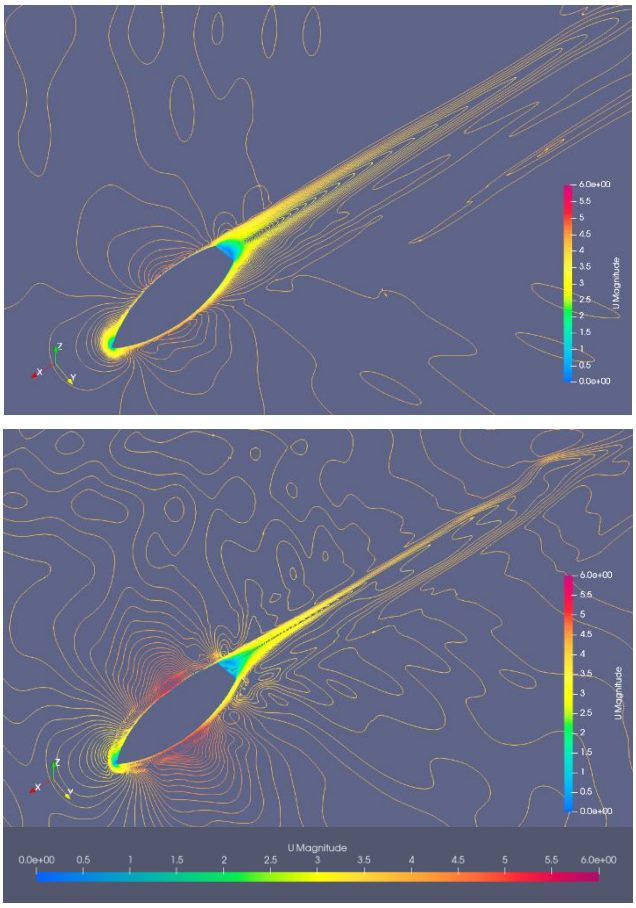


Fig.19 *Drift angle 12[deg] wake 4d and 1.2d, ISO lines*

Time for these quantitative results is around 90 days with a desktop computer: CPU Intel Core i7 6700 (Skylake), RAM DDR4 32 GB 2666 Mhz. For the same case and settings to have a medium mesh and higher accurate results, 64 GB RAM is necessary, which is confirmed with the last model set and performance for h=1.2d.

6. Conclusions

As a result of the simulations the following conclusions are made:

- can obtain free surface flow around body with good data for prediction of forces and moments acting upon hull.
- can generate quantitative results considerably fast, depending on computational resources. At least 64GB RAM is necessary for

more accurate values through medium or fine mesh compared to the coarse mesh used in this paper. However, despite this we can still get good enough results that can be of use.
- the CFD simulation with RANS enables accurate prediction flow details around hull in marginal shallow water problems and prediction of hydrodynamic capabilities for designed or an already constructed ships.

Diagrams and images from post-processing with ParaView show that the results from OpenFOAM are a reliable method for accurate prediction and analysis.

Recommendation for better understanding about ship-shape motion in shallow waters with CFD simulations is the use Multiple Reference Frame, MRF and Overset mesh techniques. MRF will recreate simulation for rotating arm test to obtain rotary derivatives, as for Overset to obtain trim and sinkage within 4DOF. It is important for harbor Pilots and ship Masters, who take sinking and trimming as a priority when navigating in channels and ports with limited depth, especially for large ships.

7. References

1. Xiao Yang, Yong Yin & Jingjing Lian (2021) A numerical study on flow field and maneuvering derivatives of KVLCC2 model at drift condition, Journal of Marine Engineering & Technology, 20:1, 52-63, DOI:10.1080/20464177.2019.1680076.

2. Maljković et al. Ship Maneuvering in Shallow and Narrow Waters: Predictive Methods and Model Development Review. Preprints2024,2024062049. DOI:10.20944/preprints202406.2049.v1.

3. Kim, D., Tezdogan, T., Incecik, A. Hydrodynamic analysis of ship manoeuvrability in shallow water using high-fidelity URANS computations. Applied Ocean Research 2022, 123. DOI:10.1016/j.apor.2022.103176.

4. Vantorre et al. Maneuvering in Shallow and Confined Water. The Encyclopedia of Marine Offshore Engineering, 2017, pp. 1–17. DOI:10.1002/9781118476406.emoe006.

5. Ivanov, P., Lefterova, M. Adoption of a digital approach for ship-shape simulation trials with CFD. Mathematical Modeling, Vol. 8 (2024), Issue 1, pg(s) 18-23. Available at: https://stumejournals.com/journals/mm/2024/1/18

6. OpenFOAM, k-omega Shear Stress Transport (SST). Available from:https://www.openfoam.com/documentation/guides/latest/doc/guide-turbulence-ras-k-omega-sst.html.

7. Menter, F.R., Langtry, R., Völker, S. Transition Modelling for General Purpose CFD Codes. Flow Turbulence Combust. 2006, **77**, 277–303. DOI:10.1007/s10494-006-9047-1.

8. Abkowitz, M.A., (1969). Stability and motion control of ocean vehicles. Massachusetts and London: MIT, Cambridge. Available at: https://mitpress.mit.edu/9780262510066/stability-and-motion-control-of-ocean-vehicles/.

9. Fossen, T. (2011) Handbook of Marine Craft Hydrodynamics and Motion Control. John Wiley & Sons, Inc., Chichester, UK. DOI:10.1002/9781119994138.

10. Ankudinov, V., Kaplan, P., Jacobsen, B.K. Assessment and principal structure of the modular mathematical model for ship maneuverability prediction and real-time maneuvering simulations. Prepared for delivery at MARSIM ’93, St. John’s, Newfoundland, Canada, September 1993. Website: https://www.imsf.org/.

11. Yasukawa, H., Yoshimura, Y. Introduction of MMG standard method for ship maneuvering predictions. Journal of Marine Science and Technology (Japan) 2015. 20: 37–52. DOI:10.1007/s00773-014-0293-y.

12. Kose, K., Misiag, W.A., Inoue K. Can one have a general, standardized maneuvering model? IMSF Workshop, Florida, USA, 1995.

13. Eloot, K. (2006). Selection, experimental determination and evaluation of a mathematical model for ship manoeuvring in shallow water. Ghent University. Faculty of Engineering, Ghent, Belgium. Available at: http://hdl.handle.net/1854/LU-757846.

14. OpenCFD Ltd. OpenFOAM, The Open Source CFD Toolbox, User Guide, 2022.

15. 2D Breaking of a dam. Available at: https://www.openfoam.com/documentation/tutorial-guide/4-multiphase-flow/4.1-breaking-of-a-dam.

16. Kleefsman, K.M.T., et al., 2005. A Volume-of-Fluid based simulation method for wave impact problems. Journal of computational physics, 206(1), 363-393. DOI:10.1016/j.jcp.2004.12.007.

17. The International Towing Tank Conference, ITTC Recommended Procedures and Guidelines. Available at https://www.ittc.info.