Adoption of a digital approach for ship-shape simulation trials with CFD

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

***Abstract**: The prediction of maneuverability is a classic problem in ship hydrodynamics. Likewise, ship motions in restricted waters have been studied for a long time. It is an indisputable fact that full-scale or model tests of the respective ship meet IMO requirements and provide the necessary information for the maneuverability of the ship in deep unconfined waters. However, they are unable to provide practical insight into understanding of ship maneuverability in shallow water, as the maneuvering behavior of a ship in such areas differs significantly. Ship maneuvering in coastal areas and harbor approaches, where space is limited, movement speeds are low, and traffic heavier requires deeper researching. The aim of the present work is to predict the hydrodynamic derivatives in the mathematical model of the maneuvering of ship in shallow waters based on CFD programs with free access like FreeCAD, OpenFOAM and ParaView.*

The sequence of solving the task is described with details in this work:

1. Generating the shape of the studied body 2. Creating the domains around the experimental body, sizes and boundaries
3. Defining the continuum, areas and subareas of study, and boundary conditions 4. Generation of the computational grid
5. Defining the continuum – physical models 6. Selection of the calculation algorithm 7. Substances modeling 8. Motion modeling.
9. Modeling of currents and energy 10. Flow turbulence modeling.

Results have been obtained for the linear hydrodynamic derivatives in the mathematical model for the maneuvering motion of a ship. They are compared with available experimental and theoretical data in the literature.

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

1. Introduction

The increasing pressure in recent years for faster creation of new and more quality products and technological processes in the industry, led to the use of numerical simulations in their development alongside the traditional experimental approach. In many cases, numerical simulations can save time and money if used as a complementary tool to the experiments.

The part of fluid mechanics dealing with simulations is called Computational Fluid Dynamics, or the commonly used abbreviation CFD. Typically, the stages that a CFD simulation goes through are problem formulation, mathematical model creation, its numerical solution, visualization and interpretation of the results. Always accompanying to the activities of each step of the simulation are the validation and verification of the used model.

2. General Setup

Right after the precise formulation of the problem of fluid mechanics, follows the construction of the mathematical model of the needed process. In the basis of all models are of momentum, mass and energy balance, which are Partial Differential Equations (PDE). All PDEs require Boundary and Initial Conditions (BC and IC) which are defined before solving them.

The final system of differential, integral and algebraic equations is too complex to be solved analytically. 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. In most CFD software packages, the discretization of the space under study is performed by the Finite Volume Method (FVM), also in some cases a practical application has the Finite Element Method (FEM).

2.1 Modelling

The basic approach in CFD modelling is to use the continuity equations for momentum, mass and energy. For this purpose, it is assumed that the variation of the quantities in each of the phases studied is continuous, i.e. the discrete nature of matter and energy is neglected. The general form of the continuity equations for any quantity φ (phi) has the form:

$$\frac{\partial \varphi}{\partial t} = -\nabla(\vec{v}\varphi) - \nabla\vec{J}_\varphi + S_\varphi$$

Where, $\partial\varphi/\partial t$ is the derivative of φ with respect to time and is equal to zero for steady motions and non-zero for unsteady motions. $\nabla(\vec{v}\varphi)$ is the convective transport term, as $\vec{v}$ is the convective velocity. $\nabla\vec{J}_\varphi$ is the diffusive (molecular) transport term, as $\vec{J}_\varphi$ is the diffusive (molecular) flux vector. Here other terms may be present also, for example in turbulent flow modelling there are additional terms with the same structure. They are treated by the computational algorithm in the same way but have a different physical meaning. And finally, S_φ is a source term that largely depends on the type of quantity φ, as in the momentum balance equation these are forces.

In this article further assumption has been made, like the effect of temperature is not considered and the fluid is treated as isothermal. Therefore, we will not deal with the internal energy conservation law, but only with mass and momentum conservation.

- ***Continuity Equation***

A fundamental physical law is that of conservation of mass, from which the so-called continuity equation is derived:

$$\frac{\partial \rho}{\partial t} = -\nabla(\rho\vec{v})$$

Where, the density ρ is a transport quantity.

- ***Motion Equation (Navier-Stokes Equation)***

The fluid flow velocity is a fundamental quantity in CFD and it is defined by the Navier-Stokes equation:

$$\frac{\partial(\rho\vec{v})}{\partial t} = -\nabla(\rho\vec{v}\vec{v}) - \nabla\bar{\bar{\tau}} - \nabla p + \rho\vec{g}$$

In the motion equation, the transport quantity is the momentum $\rho\vec{v}$. The stress tensor $\bar{\bar{\tau}}$ is the diffusion term and is a function of the velocity gradient. For a Newtonian fluid, the tensor has the form:

$$\bar{\bar{\tau}} = \mu[\nabla\vec{v} + (\nabla\vec{v})^T] + (k - \frac{2}{3}\mu)(\nabla.\vec{v})\delta$$

Where, μ is the dynamic viscosity coefficient which, in Newtonian fluids, is independent from tangential stress τ and time t.

The last two terms in the equation are the pressure gradient p and the force of gravity ρg respectively.

The joint solution of the two equations, the continuity equation and the equation of motion, gives results for the values of pressure and velocity, which must satisfy both equations. Very often the solution scheme involves a two-step procedure: first a value for the pressure gradient is given and with this value the velocity is calculated, then a pressure correction is made and iteratively the values of pressure p and velocity v are obtained. The scheme is known as the predictor-corrector algorithm.

2.2 Boundary and Initial conditions

The partial differential equations in the model cannot be solved without boundary and initial conditions.

The initial conditions represent values or functional dependence of the transport quantity φ at the beginning of the simulation for all control elements.

There are three main types of BCs:

- Boundary condition of the first-type (*Dirichlet boundary condition*) is a given value of φ on the corresponding boundary.
- A boundary condition of the second-type (*Neumann boundary condition*) is a given value for the derivative of φ along the normal to the boundary surface.
- Mixed boundary condition (*Cauchy boundary condition*) is a combination from the value of φ and its derivative on the boundary surface.

BCs which are mostly used are:

- *Inlet* - here the values of all transport quantities are set at the inlet of the computational domain. Also, functional dependencies can be specified.
- *Outlet* - in this type of boundary condition the pressure is usually set and the velocity is calculated.
- *Wall* - most modelled systems are surrounded by a rigid wall, which can be movable or fixed. Walls are assumed to have no mass transfer through them but can conduct heat. For this purpose, the velocity component normal to the wall surface is assumed to be zero for fixed walls and equal to the wall velocity for moving walls. For no-slip walls the tangential velocity component is assumed to be zero.
- BC for *symmetry* - when symmetry exists with respect to a surface, only one symmetric half of the object is considered. In order to satisfy the BC symmetry, not only geometric symmetry but also physical symmetry is required with respect to all transport quantities φ. All gradients normal to the symmetry surface are zero, as are the velocity components normal to the symmetry surface.

2.3 Turbulent flows

Turbulent flows are characterized by eddy structures of variable magnitudes that cause high frequency fluctuations to all transport quantities. The great difficulty in solving turbulence problems is that the use of grids with very small cells and very small-time step is necessary to capture the high frequency fluctuations. According to Wilcox [3], the number of computational points in Direct Numerical Simulation (DNS), N_{DNS} is of too high order:

$$N_{DNS} = (0.088R_e)^{9/4}$$

Where, R_e is the Reynolds number for the corresponding flow.

In most cases considering practice, the exact structure of the turbulent flow is not of that much interest, while the local averaged value of the quantities is more important. Currently, the following two approaches are used to solve the problem: one is with averaging the quantities over time, called RANS (Reynolds-averaged Navier-Stokes), and the other is called LES (Large Eddy Simulation).

In RANS, all transport quantities are averaged over all turbulent fluctuations in time, which allows the use of relatively coarse grids and does not require large computational resources. The influence of turbulence is accounted for by additional equations to the model, the so-called turbulence models.

In LES, a grid of a size that will allow the direct simulation (as in DNS) of large turbulent structures (whirlwinds = eddy's) is used, while for small ones a modelling approach similar to RANS is used. The modelling of small turbulent structures is called SGS modelling (Subgrid-Scale modelling). An important value in LES is the magnitude of the filter G (filter function), determining which structures are large and which are small. This value depends on the grid size, so that the user can change it indirectly.

2.4 RANS

The full Navier-Stokes equations are used with very high accuracy in laminar steady flows. For turbulent flows the equations in their averaged form are used.

Reynolds-averaged means representing all time-dependent variables in the form of two components: a time-averaged principal component $\overline{\varphi}$ and a fluctuating component, φ', i.e.

$$\varphi = \overline{\varphi} + \varphi'$$

This yields the averaged equations of the model:

$$\nabla \overline{\vec{v}} = 0$$

$$\rho \frac{\partial \overline{\vec{v}}}{\partial t} = -\rho \overline{\vec{v}} \nabla \overline{\vec{v}} - \nabla \overline{\vec{\tau}}_t + \mu \nabla^2 \overline{\vec{v}} - \nabla \overline{p} + \rho \vec{g}$$

In the second equation appears an additional term $\nabla \overline{\vec{\tau}}_t$, called the Reynolds stress tensor.

With the inclusion of this additional term, the system of equations becomes overdetermined. To overcome this problem, there are several approaches to model the stress tensor. In one approach, additional models, called turbulence models, are added to the system of equations.

The turbulence models used with the time-averaged equations are the following types, mentioned briefly:

1. Length Mixing model
2. Single-equation or "one-parameter" models
3. Two-equation ($k-\varepsilon$ and $k-\omega$ models) or "two-parameter" models
4. Reynolds stress model
5. Algebraic stress model

The first three groups of models are known as "Eddy (turbulent) viscosity models" and are typically based on the most commonly used approach, known as the Boussinesq approximation (hypothesis) for the relationship between the derivatives of the averaged velocities and the Reynolds stresses.

When considering flow around body circulation cases characterized by high values of Reynolds numbers, it is recommended that the averaged equations of motion to be used with the Spalart-Allmaras and Menter turbulence models. These turbulent models have various modifications that are given below in connection with the particularities of their use as a function of the values of the dimensionless parameter y^+, representing the local Reynolds number in the region of contact:

$$y^+ = \frac{y.u_+}{\nu}$$

Where, y is distance to the wall,

$u_+ = \sqrt{\frac{\tau_\omega}{\rho}}$ is friction velocity near walls,

τ_ω is tangential stress to the wall,

ν is kinematic viscosity coefficient.

Spalart-Allmaras models	Model for considering areas near walls
Standard	- for low parameter values y^+ (Low y^+ Wall Treatment) - for all parameter values y^+ (All y^+ Wall Treatment)
High-Reynolds Number model	For high parameter values y^+ (High y^+ Wall Treatment)
DES (Detached Eddy Simulation)	- for low parameter values y^+ - for all parameter values y^+

Menter models, K-Omega	Model for consideration areas near walls
Standard	Menter's models can be applied to all cases of parameter values y^+, respectively: - at low values, i.e. $y^+ \to 0$ - at high values: $y^+ \geq 30$ - for all values of y^+
BSL (baseline) K-Omega	
SST (shear-stress transport) K-Omega	
SST K-Omega DES (Detached Eddy Simulation)	

2.5 Numerical procedure

The system of integrodifferential and algebraic equations can be solved analytically for only a small number of cases. For all other important practical cases, a computer-based numerical solution is necessary. The basic idea of the numerical solution is to replace the PDE, which is continuously defined in time and space, by a system of algebraic equations that gives a solution only for predefined discrete points or discrete intervals. For the Finite Difference Method (FDM) values are calculated for individual points, for the FVM values are obtained that are constant over the entire volume of the cell, and for the FEM the coefficients of the polynomial function that yields the solution for the corresponding element are obtained. All of these solutions are for each discrete point (period) in time. In order to obtain such a numerical solution it is first necessary to perform a discretization of the system in time and space. Time is a one-dimensional independent variable and is simply divided into discrete intervals called a time step.

The time step can be either constant or variable for the duration of the simulation. The choice of the time step size depends on the desired accuracy of the solution, the robustness of the computational procedure (these two criteria require a decrease in the time step) and the time required to solve the problem (usually an increase in the size is requested), furthermore in some computational schemes the time step size is also tied to the grid size. The discretization of the space in the two most used in CFD approaches FVM and FEM, requires the partitioning of the whole study area into smaller non-overlapping cells, which together form a so-called grid.

Grids are divided into structured grid and unstructured grid. In a structured grid, the individual faces of the cells are parallel to each other and can be uniform and non-uniform. The shapes of a cell for the structured grid are of six-faces with four-edges in a two-dimensional grid (hexahedral mesh). This type of mesh has limited application for complex geometries because it is very difficult (and sometimes impossible) to create, but for simple geometries it is relatively easy to generate. In unstructured meshes, the sides of the cells intersect and the shapes are irregular, making them preferable for complex geometries, but on the other hand their automatic generation is difficult. The shapes of unstructured meshes are of six-sided polygon (hexagons), triangular prisms (wedges), four-edge base and triangular faces with apex (pyramids) or a polyhedron composed of four-faces, six straight edges and four-vertices (tetrahedrons), and in this two-dimensional case they are quadrilaterals or triangles. The most common requirements for meshes are:

- geometric - the outer boundaries of the mesh should reproduce the geometry as well as possible in order to set the boundary conditions correctly.
- physical - to obtain a solution with the desired accuracy, the grid needs to be finer in areas where there is a sharp change in gradients and in areas where the gradients are not constant but variable.

2.6 Discretization in time

In non-stationary processes, the time derivative is non-zero and the following two approaches to obtaining a solution are mainly applicable:

- explicit method - this method is easy to implement because a system of independent linear algebraic equations is obtained, and it is accurate to the first order and has conditional stability.

With it, the value of φ at the moment $t + \Delta t(\varphi_p^{t+\Delta t})$ depends only on values from the previous moment of time t. For convective processes (as are most processes modeled by CFD) it is necessary to observe the so-called Courant-Friedrich-Levy condition:

$$C_\Delta = \frac{|v|\Delta t}{\Delta x} \leq 1$$

Where, C_Δ is Courant number.

- implicit method - the method is first-order accurate and unconditionally robust.

With it, the value of φ at the moment $t + \Delta t(\varphi_p^{t+\Delta t})$, depends on values from the previous moment of time t as well as from the values on the surfaces of the control volume at the current moment $t + \Delta t(\varphi_p^{t+\Delta t})$.

2.7 Multiphase flows with free surface

Free surface is modeled using the Volume of Fluid approach (VOF) with a High-Resolution Interface Capturing Scheme. The numerical model can be applied to any structured and unstructured grid with arbitrary shaped CV. With two conservation and one turbulence equations an additional equation is solved, introducing a variable α ($0 \leq \alpha \leq 1$) for the volume fraction of each fluid, where 1 stands for filled CV of the fluid in which at the same time the volume fraction of other fluid has to be 0 to achieve unity. This approach treats both fluids as a single fluid which changes its properties according to the volume fraction.

3. Ship-shape Simulation with CFD

The analyses were performed using the software product OpenFOAM version 2212. It was chosen due to the fact that the software package is with free access, unlike most similar CFD products such as STAR-CCM+, Ansys-Fluent and others which are distributed commercially.

3.1 Open∇FOAM Description

OpenFOAM is distributed by the OpenFOAM Foundation, an organisation holding the copyright of the software and its documentation. The purpose of the Foundation is to manage and distribute OpenFOAM as free open-source software for the benefit of its users. It is a registered company limited by guarantee based in England. OpenFOAM® is a registered trademark of OpenCFD Ltd, licensed to the OpenFOAM Foundation in 2011 [4].

Open-Source Field Operation and Manipulation (OpenFOAM®) is a framework for developing executable applications based on C++. It consists of 250 pre-built applications which can be divided into *solvers* and *utilities*. Solvers are designated to solve a specific problem in fluid (continuum) mechanics. Utilities are intended to perform tasks that involve data manipulation.

FreeCAD v0.21.1 workbench cfdOF v1.24.1 [5] is used as front-end graphical user interface, GUI for the ESI-OpenFOAM [6]. Selecting the relevant physics, environment properties, generating mesh, assigning boundary conditions, and the chosen solver settings are specified within.

3.2 Generating shape of the studied body

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

***Table 1**: Boat model particulars*

		Design
Length of waterline	L_{WL}	41.21 m
Breadth	B	15.74 m
Draft	d	3.27 m
Wetted Surface Area	S_0	288.65 m^2

The process of generating can be performed using *Sketcher* workbench to draw 2D shape and *Part Design* workbench to transform the drawing into 3D model. Shape modifications like scaling or changing coordinates are possible in late steps from *Draft* workbench. The chosen scale is one to one, full scale because of viscous forces (ITTC, 1978) [7].

There is an option to use complex geometry of a ship-shape body to be imported directly from other projects. The software program supports different types like *stl*, *igs*, *stp* etc. that are used for export from other modern CAD products. Files as *stp* or *iges* are preferable because they are not large in size and contain information with extreme precision and high levels of detail compering to other that describe a model's exterior geometry and simplify its features into a mesh made of triangles that's free of gaps.

Importing a such low-quality body requires additional processing because of its simplicity data. *Mesh* workbench can manipulate and convert the shape to fine watertight model, then in *Part* workbench the mesh is converted to solid. To continue further with the study object settings and achieve engineering results, the surface should be with high precision and suitable sharpness of surface edges (Fig. 1).

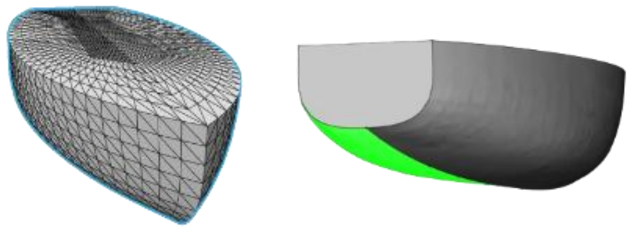

***Fig. 1** Body shape evolution*

3.2 Domain dimensions and boundary conditions

OpenFOAM finite volume method uses a co-located methodology on an unstructured polyhedral grid with arbitrary grid elements. Fluid dynamic quantities are centred on the control volume centroids. 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, only half of the modelled is computed. The domain length is taken up to seven model lengths, two lengths in front and four lengths behind the hull. Furthermore, three model length wide and half model length above. Depending on draft there is a free surface at 4d distance from the bottom, as main definition for deep water (Fig. 2).

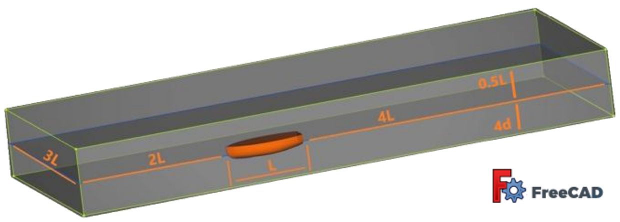

***Fig. 2** Domain*

The boundary condition applied to the model hull is a no-slip wall allowing for boundary layer around the model hull to develop. y^+ wall treatment with ten layers is used for a hull modelling assumption. Bottom and side boundaries are modelled as symmetry boundaries, while top is open boundary for case of deep water. As for shallow water case, bottom boundary condition is changed to no-slip wall. The velocity inlet and the pressure outlet are applied to inlet and outlet boundaries (Fig. 3).

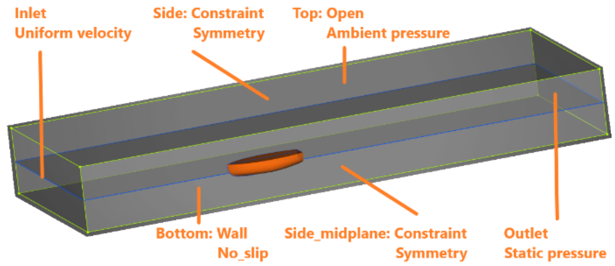

***Fig. 3** Fluid BCs Sub-types*

3.3 Generation of the computational grid

It is well known that the mesh is of heist importance in the 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 for mesh is in must be done check list task for obtaining accurate results and to score a mesh with OK test for topological errors or orthogonality, skewness, aspect ratio and smoothness.

A utility called cfMesh is an open-source library for mesh generation implemented on top of the OpenFOAM framework and cell type with polyhedral form were chosen for this work (Fig. 4).

Base cell sizes in the domain are gradually refined towards the regions where high gradients of velocity and pressure are expected with minimum of three layers of same size till next level. A prismatic layer is generated around the hull model, defining the appropriate cell thicknesses in order to resolve the boundary layer.

Cell sizes, refinements and additional layers to body were chosen for optimization of time and mesh quality that will accomplish OK test according case complexity and domain dimensions. Parameters used in the generation and obtaining data are identical for all velocities with some change for drift angles and shallow water conditions (Tab. 2a).

Created additional hull and free surface mesh refinement of 1m, with relative element size 0.5 to base cells, added ten boundary layers with expansion ratio of 1.2 with max first cell height to 0.2 m. Check mesh results after grid generation (Tab. 2b).

***Table 2a:** Mesh Quality Criteria by checkMesh in OpenFOAM*

Mesh Characteristics	Criteria
Max. Cell Non-orthogonality [deg]	70
Max. Cell Skewness (Internal)	4
Max. Cell Skewness (Boundary)	20

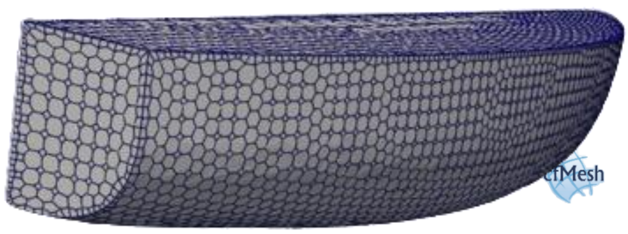

***Fig. 4** Half surface hull mesh generated for CFD purpose.*

***Table 2b:** Grid Quality of Reference Case, e.g. 5deg, h=18*

Mesh Characteristics	Values
Cells in millions	2 830 515
Max Aspect Ratio	42.71
Max Non-orthogonality [deg]	73.73
* faces > 70 degrees	2
Avg Non-orthogonality [deg]	4.62
Max Cell Skewness	2.64

3.4 Defining the continuum - physical models

The physical models chosen for the simulation are:

- incompressible fluids
- transient motion
- turbulent flows
- multi-phase: seawater and air
- immiscible fluids
- VOF
- isothermal fluids

InterFoam solver for two incompressible, isothermal, immiscible fluids, using a VOF phase-fraction based interface capturing approach is used to solve the governing equations of continuity and momentum in a viscous flow, around the hull model.

The Menter Shear Stress Transport Model (SST) k-ω turbulence model [8] is used in the simulations, in which the transport equations are solved for two variables: turbulent kinetic energy k and specific dissipation rate ω [9].

3.5 Numerical analysis

Numerical studies were conducted for four cases with various ship speeds corresponding to Froude number of 0.12, 0.16, 0.20 and 0.26, with different water depth h and setting relative depths for h/d to 4.0 (deep water), 1.5 (moderate shallow water) and 1.2 (marginal shallow water).

To simulate different depths of the water basin, the computational domain dimensions are changed - such is the distance between the waterline and the bottom. In the case for shallow water, the BC for the computational domain of the bottom is changed to no-slip wall.

For non-stationary numerical simulation, it is necessary to set the following parameters:

- time step
- maximum number of internal iterations
- physical time (simulation period)

The cases in this work were solved with a time step set to 0.01 sec, a maximum number of internal iterations 20, and a simulation time of 5 sec.

The computation process is automatically visualized by the residual difference plot, giving information about the convergence of the obtained solution.

In Fig. 5 is presented analogous graphic for a case of boat motion in deep water conditions with speed 5.2 m/s.

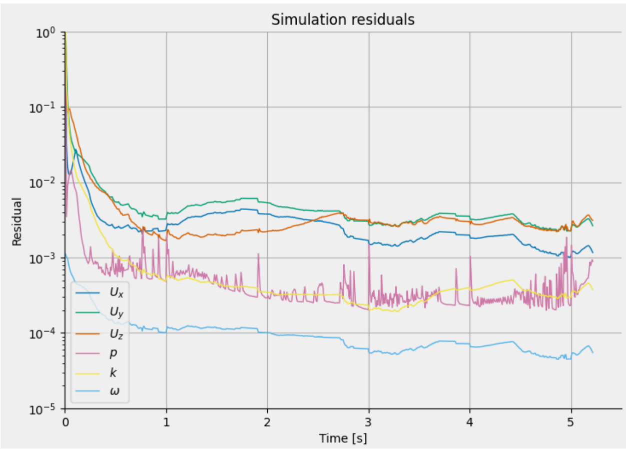

Fig. 5 *Residual plot*

Illustrations of distribution for water and air phases on the lateral hull projection, respectively at the initial movement moment (standstill) at Fig.6a and at a moment of progressive movement Fig.6b of the boat.

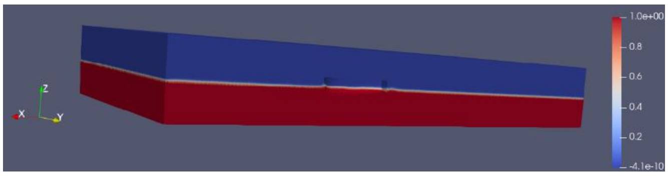

Fig. 6a: *Fluid phases of the domain*

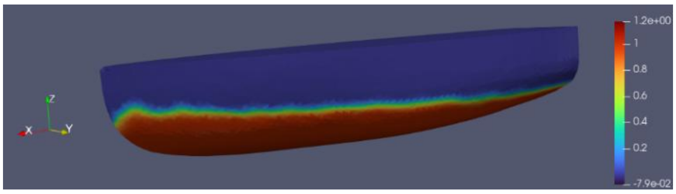

Fig. 6b: *Fluid phases on body*

The numerical studies were performed on a personal computer Intel® Core™ i7-6700 CPU@3.40GHz with 32 GB RAM.

3.6 Numerical results

The results obtained for the flow circulation structure, velocity variations and forces from pressure and friction are given as images of velocity vectors, scalar characteristics, current lines and iso surfaces. All post-processing data images are visualized with ParaView [10].

Estimates of the flow structure

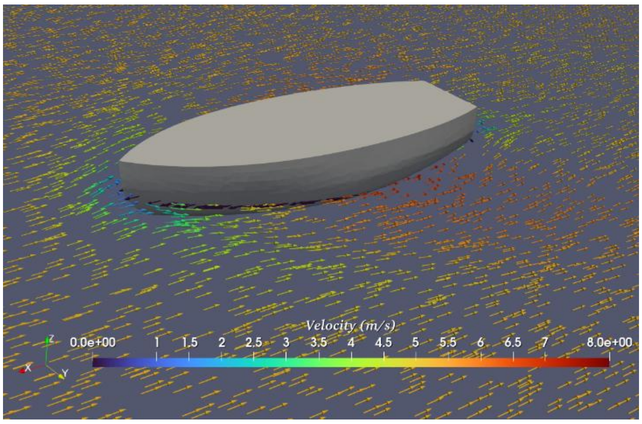

Fig. 7 *Free surface speed vectors flow around boat, initial speed* $v = 5.2\ m/s$

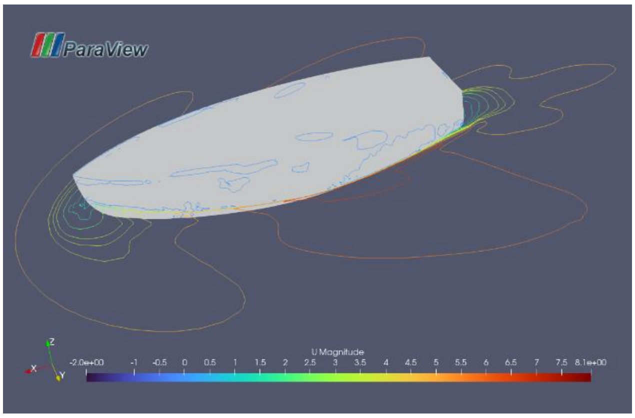

Fig. 8: *Isolines of velocity flow around hull on the undisturbed water surface for shallow water* $h/d = 1.2, v = 5.2m/s$

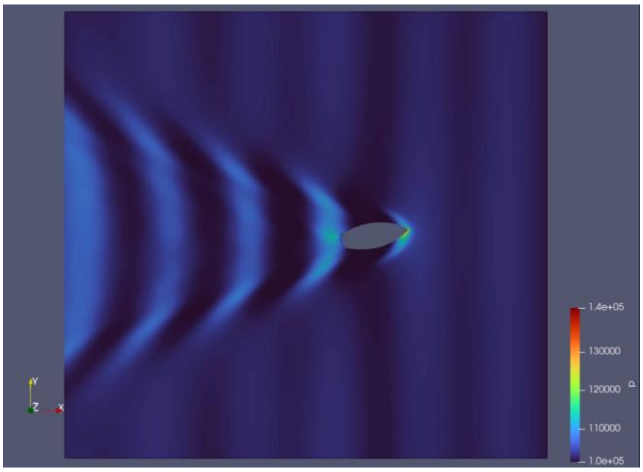

Fig. 9a: *Wave pattern around hull in deep water, h/d = 4.0 (boat motion with 5° drift angle)*

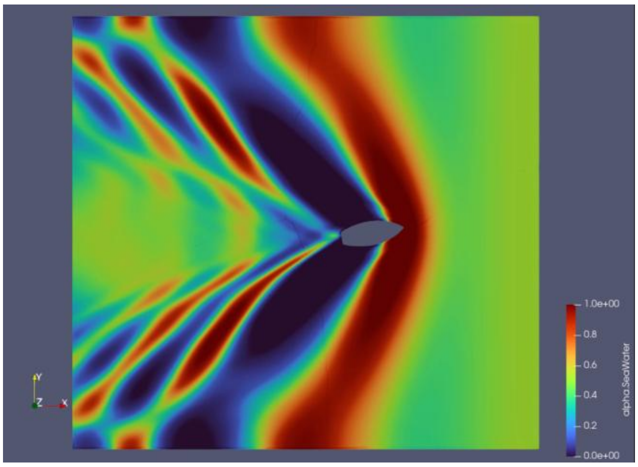

Fig 9b: *Wave pattern around the hull in marginal shallow water, h/d = 1.2 (boat motion with 5° drift angle)*

As a conclusion, it is obvious that in the case of 'deep water' the wave pattern resembles the classical Kelvin wave system known from the ship theory. Shallow water (corresponding to a critical flow around ship-shape model with $F_{n_h} = 0.45$, $h = 4\ m$) leads to a complete disappearance of the transverse wave system and the formation of an intense bow wave with an angle larger than the Kelvin wedge.

Estimates of change in pressure and friction forces

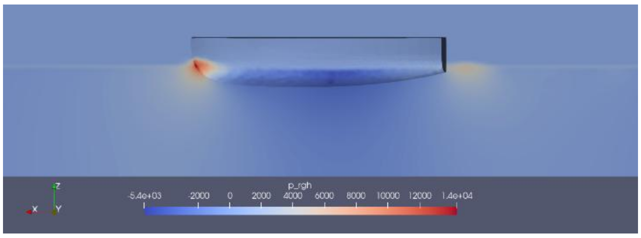

Fig. 10: *Pressure variation around hull in deep water, h/d = 4.0*

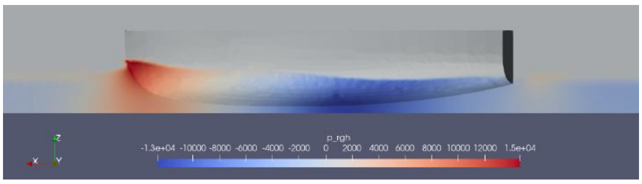

Fig. 11: *Pressure variation around hull in marginal shallow water, h/d = 1.2*

4. Conclusions and Recommendations

As a result of the research the following conclusions can be drawn:

- a large number of numerical studies have been carried out and the adaptation of the OpenFOAM software package for solving problems related to the hydrodynamic characteristics of ship-like bodies has been achieved;

- the results of the numerical studies are subject to further analysis and synthesis, which will be the subject of the dissertation and future publications;

- the described methodology provides a good basis for future research;

- the problem of motion for a ship-like body onto free surface of the fluid allows to obtain the corresponding coefficients of both hydrodynamic and aerodynamic effects. In doing so, it is possible to introduce laws of change for velocity circulation flow around the hull for both fluids.

5. References

1. Лефтерова, М. Изследване на приложимостта на CFD в динамиката на флуидите на основата на анализ на обтичането на тестово тяло. *Годишник на Технически университет, Варна*, 2010.

2. Nakayama, Y., Boucher, R.F. Introduction to fluid mechanics. *Butterworth-Heinemann*. 1999, ISBN:0340676493.

3. Wilcox, D. Turbulence Modeling for CFD. *DCW Industries, Inc., La Canada, CA, third edition*, 2006.

4. OpenFOAM Foundation. Available from: https://openfoam.org/.

5. FreeCAD workbench CfdOF. Available from: https://github.com/jaheyns/CfdOF.

6. ESI OpenFOAM. Available from: https://www.openfoam.com/.

7. The International Towing Tank Conference. Available from: https://ittc.info/.

8. 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.

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

10. ParaView. Available from: https://www.paraview.org/.