Scientific Journal of Silesian University of Technology. Series Transport
Zeszyty Naukowe Politechniki Śląskiej. Seria Transport

Volume 110 **2021**

p-ISSN: 0209-3324

e-ISSN: 2450-1549

DOI: https://doi.org/10.20858/sjsutst.2021.110.5

Journal homepage: http://sjsutst.polsl.pl

Article citation information:
Illes, L., Jurkovic, M., Kalina, T., Gorzelanczyk, P., Luptak, V. Methodology for optimising the hull shape of a vessel with restricted draft. *Scientific Journal of Silesian University of Technology. Series Transport*. 2021, **110**, 59-71. ISSN: 0209-3324.
DOI: https://doi.org/10.20858/sjsutst.2021.110.5.

Ladislav ILLES[1], Martin JURKOVIC[2], Tomas KALINA[3], Piotr GORZELANCZYK[4], Vladimir LUPTAK[5]

METHODOLOGY FOR OPTIMISING THE HULL SHAPE OF A VESSEL WITH RESTRICTED DRAFT

Summary. Increasing transport volumes on Europe's inland waterways is a major reason for improving the quality and reliability of internationally important waterways. Continued navigation restrictions due to restricted draft (draught) led to the search for new design solutions. Such solutions enable navigation even under critical navigation conditions. Restricted draft is one of the most important limitations that hinder navigation, especially in the summer. The main objective of the construction of an inland vessel is to obtain a shape that will achieve optimum performance with as little resistance as possible. A shape that will be able to navigate at a limited depth. Presently, there is no clearly defined methodology as a procedure for optimising the hull. When solving theoretical problems of shipbuilding character and ship calculations, it is necessary to consider the basic

[1] MULTI engineering services, Dunajske nabrezie 4726, 94501, Komarno, Slovak Republic.
Email: ladislav.illes@multi.engineering. ORCID: https://orcid.org/0000-0002-8502-4719
[2] Faculty of Operation and Economics of Transport and Communication, University of Zilina, Univerzitna 1, 010 26 Zilina, Slovakia. Email: martin.jurkovic@fpedas.uniza.sk.
ORCID: https://orcid.org/0000-0001-7673-1350
[3] Faculty of Operation and Economics of Transport and Communication, University of Zilina, Univerzitna 1, 010 26 Zilina, Slovakia. Email: tomas.kalina@fpedas.uniza.sk. ORCID: https://orcid.org/0000-0003-0564-086X
[4] Stanislaw Staszic University of Applied Sciences in Pila, Polytechnic Institute, Podchorążych 10 Street, 64-920 Piła, Poland. Email: piotr.gorzelanczyk@pwsz.pila.pl. ORCID: https://orcid.org/0000-0001-9662-400X
[5] Department of Transport and Logistics, Faculty of Technology, Institute of Technology and Business in Ceske Budejovice, Czech Republic. Email: luptak@mail.vstecb.cz. ORCID: https://orcid.org/0000-0001-7550-5714

theory of the ship with special regard to the latest methodological procedures of related scientific disciplines. This paper presents a methodology that considers all the basic aspects of optimisation tasks in ship design and construction.

Keywords: optimisation, methodology, restricted draft, waterways, vessel

1. INTRODUCTION

Shipping on European rivers and inland waterways is of considerable economic benefit, contributing to the carriage of goods and passenger transport. The most important waterway is the Rhine-Main-Danube Canal, which connects the North Sea with the Black Sea. However, shipping on some significant critical stretches of the Danube has been restricted by long-standing problems of limited seasonal navigation conditions. Sometimes the traffic of large commercial vessels and pushed convoys is completely paralysed [9,11].

The flow regime and water level has been developing and changing globally for a long time. This is due to anthropogenic effects such as water management operation or land use change near waterways. This has an impact on freshwater resources and changes in hydrological conditions that form the basis for water resources and the entire ecosystem [5].

Because of rapid climate change in recent years, there have been serious seasonal problems on the Rhine, where it has become necessary to restrict freight transport to such an extent that it has caused difficulties in the logistics systems of large companies that mainly source their raw materials and semi-finished products through shipping [2]. These changes represent a global problem for navigation on inland waterways, water areas and ports and have become major obstacles to the further development of water transport [16,19].

Traffic restrictions due to the low water level can similarly be seen on the Danube River. In January 2017, the water levels were extremely low, around the lowest navigable water level (LNWL), when the river froze over, and subsequently, ice events occurred. Ship traffic on the Upper Danube and the Middle Danube saw a considerable decline in loaded drafts during this period. After measures to combat the ice events on the Danube had been concluded and navigation conditions stabilised, operation at loaded drafts of approx. 2,5 m for barges in pushed convoys began in early March. Hydrological conditions on the Danube were unstable during the second quarter, and by late May, loaded drafts were no more than 2,3-2,2 m. [4]. The summer low-water period began in June, and successive, intermittent precipitation in the third and fourth quarters did not lead to a stabilisation of hydrological conditions, hence, loaded drafts remained between 2, 2 and 2, 3 m from September until the end of the year 2017 [3].

The low water level lasted for 80 days, resulting in the cessation of navigation and the associated losses for shipowners [1]. Optimising the shape of the vessel's hull is one way to improve shipping operations on inland waterways. Particularly, the improvement of the navigability characteristics of vessels navigating in shallow waters. Restricted draft is one of the most important limitations that reduces navigation, especially during summer.

An inland vessel floating in shallow water achieves a much higher resistance than a vessel floating in deep water. There are several methods by which these effects can be eliminated [8,15,21,25,27]. In addition to resistance, wake fraction and thrust deduction change as well [20,23]. The propulsion efficiency likewise varies depending on the different propeller load [12]. Optimisation of ship hull–propeller system is also one of the most important aspects of ship design and leads to ship cost reduction, improving performance and increasing the lifespan of the propulsion system. Changes in the stream of water surrounding the vessel are caused by different flow around the hull compared to deep water navigation [7]. The low ship underneath below the vessel leads to increased return flow speed. In the vertical direction, the movement of water is limited, causing it to be transformed into a horizontal movement. Thus, the increased reverse flow rate reduces the pressure below and around the hull. This results in an additional reduction in the draft and, in most cases, increased wave resistance [13,26].

The specific effects that change the trajectory of liquid flow around the ship hull in shallow water require different design requirements compared to seagoing ships. Inland vessels often navigate in waters with a depth of about 2 m. The main objective of the design of shallow-water vessels is to achieve the shape of the hull, with the lowest propulsion power requirements. Navigation optimisation studies, focusing on the impact of waves on shallow watercraft, have been conducted in the past [22,24,29].

This paper is based on the objectives set out in the Danube Strategy and follows up the scientific publications on the research and development of new and innovative types of ships and propulsion vehicles designed for the changing conditions of the Danube navigation. However, the results are general, globally applicable on inland waterways with similar parameters. The result of this paper is a methodology of optimising the shape of the ship hull with restricted draft, which will be the basis for the application part with a specific proposal of solution.

2. THEORETICAL BACKGROUND AND METHODS

In solving the theoretical problems of shipbuilding character and naval architectural calculations, the basic theory of ship will be applied with special regard to the latest published results of related scientific disciplines.

2.1. Equations describing fluid flow

When examining fluid flow, the basic laws of physics apply, that is, conservation law of mass, conservation law of momentum and conservation law of energy. All these laws, as well as viscous phenomena in real fluid, are reflected in Navier-Stokes equations that describe both laminar and turbulent flow.

For incompressible fluids, where ρ = const., and $\frac{\delta\rho}{\delta t} = 0$, the continuity equation in a differential form will be:

$$\frac{\delta v_x}{\delta x} + \frac{\delta v_y}{\delta y} + \frac{\delta v_z}{\delta z} = 0 \qquad (1)$$

When a hexahedron represents the liquid particle, its centre of gravity will be affected by the mass forces X, Y and Z in the corresponding directions. Surface forces from external pressure will act in the centre of gravity of the elementary body surfaces.

The hydrostatic Euler equations in the state of flow will have nonzero values on the right side. These will be x, y, and z acceleration components that express forces per unit mass.

Hydrodynamic Euler equations for the ideal fluid (when viscosity is not considered) are then obtained by substituting the inertia forces generated by the acceleration of the fluid particles into the equations:

$$X - \frac{1}{\rho} \cdot \frac{\delta p}{\delta x} = a_x$$

$$Y - \frac{1}{\rho} \cdot \frac{\delta p}{\delta y} = a_y \qquad (2)$$

$$Z - \frac{1}{\rho} . \frac{\delta p}{\delta z} = a_z$$

whereby adjusting the accelerations on the right side, we get Euler's partial differential equations that express the dependence between unit mass, surface and inertia forces:

$$\begin{gathered} X - \frac{1}{\rho} \cdot \frac{\delta p}{\delta x} = v_x \cdot \frac{\delta v_x}{\delta x} + v_y \cdot \frac{\delta v_x}{\delta y} + v_z \cdot \frac{\delta v_x}{\delta z} + \frac{\delta v_x}{\delta t} \\ Y - \frac{1}{\rho} \cdot \frac{\delta p}{\delta y} = v_x \cdot \frac{\delta v_y}{\delta x} + v_y \cdot \frac{\delta v_y}{\delta y} + v_z \cdot \frac{\delta v_y}{\delta z} + \frac{\delta v_y}{\delta t} \\ Z - \frac{1}{\rho} \cdot \frac{\delta p}{\delta z} = v_x \cdot \frac{\delta v_z}{\delta x} + v_y \cdot \frac{\delta v_z}{\delta y} + v_z \cdot \frac{\delta v_z}{\delta z} + \frac{\delta v_z}{\delta t} \end{gathered} \tag{3}$$

Derivatives according to *x*, *y* and *z* on the right side of the equations express the acceleration components along the curved streams. The last derivative by *t* is a local component that expresses the change in velocity over time.

Considering both the flow viscosity of the fluid, and the corresponding shear forces would also act on the walls of the elemental hexahedron - in addition to the compressive forces.

Newton's equation expresses these frictional forces mathematically as tangential stress

$$\tau = \mu \cdot \frac{dv}{dx} \tag{4}$$

where η is the coefficient of dynamic viscosity.

By adding the friction component to the Euler equations of fluid dynamics, we obtain the Navier-Stokes partial differential equations for all unit actions on the fluid particle in three basic directions of space, that is, weight, pressure, friction and inertia. Furthermore, considering the continuity equation (1), the Navier-Stokes equations expressing the flow of Newtonian fluid can be further simplified to the form:

$$\begin{gathered} X - \frac{1}{\rho} . \frac{\delta p}{\delta x} + v . \left(\frac{\delta^2 v_x}{\delta x^2} + \frac{\delta^2 v_x}{\delta y^2} + \frac{\delta^2 v_x}{\delta z^2} \right) = a_x \\ Y - \frac{1}{\rho} . \frac{\delta p}{\delta y} + v . \left(\frac{\delta^2 v_y}{\delta x^2} + \frac{\delta^2 v_y}{\delta y^2} + \frac{\delta^2 v_y}{\delta z^2} \right) = a_y \\ Z - \frac{1}{\rho} . \frac{\delta p}{\delta z} + v . \left(\frac{\delta^2 v_z}{\delta x^2} + \frac{\delta^2 v_z}{\delta y^2} + \frac{\delta^2 v_z}{\delta z^2} \right) = a_z \end{gathered} \tag{5}$$

Equations 5 can be interpreted as the specific form of the second Newton's law for the flow of viscous incompressible fluid per unit mass, on the right with the product of acceleration and weight, on the left with the sum of mass and surface (pressure and viscous) forces [6,10].

2.2. Computational domain and CFD mesh

Computational Fluid Dynamics (CFD) is the most commonly used software in computer modelling of fluid flow. Several mesh-based methods have been developed in this area where the geometry under investigation is replaced by a 2D or 3D mesh and the flow problem is solved using Navier-Stokes equations. The basic principle of CFD is to create a computational domain that consists of a geometric model of the actual and discretised form (mesh), a definition of boundary conditions, a set of physical properties and calculation methods, and possibly external geometry boundaries of the flow area (external flow) [17,28].

In the CFD simulation of navigation, the geometry under investigation consists of the outer surface of the hull, surrounded by a flow area, mostly of hexahedral shape. This is a typical case of external flow where the flow takes place in the surrounding environment and not within the computational geometry. The investigated physical phenomena take place in a multiphase environment, at the boundary of two phases (water-air), which considerably increases the computational complexity of these tasks.

The resulting mesh is a product of discretisation of real geometry, its arrangement can be either structured or unstructured.

Structured mesh consists of rectangular elements (in 2D) and hexahedral blocks (in 3D). The main advantage of such a mesh is higher accuracy of calculation and less complex matrix representation of the solved structure within the algorithm. However, the discretisation of complicated shapes and the creation of transition areas with different meshing size often give rise to problems that point to less flexibility in the structured mesh.

In some cases, an unstructured mesh with sufficient accuracy can be similarly used. More so, there are tasks that do not allow the application of structured meshes. Such a mesh consists of triangular elements (in 2D) or tetrahedral, prismatic pentahedron and pyramidal elements (in 3D). It is ideal for discretising complex geometric shapes, maintaining good quality in shape interpolation (small distortions), and densifies without problems. The various elements are often combined into an optimal structure, for example, in the boundary layer zone.

Another possibility is the creation of hybrid structures, which is a suitable combination of structured and unstructured meshes. It has wide application in CFD simulations, where the complex surfaces of bodies and their boundary layers are represented by an unstructured mesh, while the environment is formed by a high-quality structured mesh. The interface zone between them is a transition area filled with pyramidal elements (in 3D).

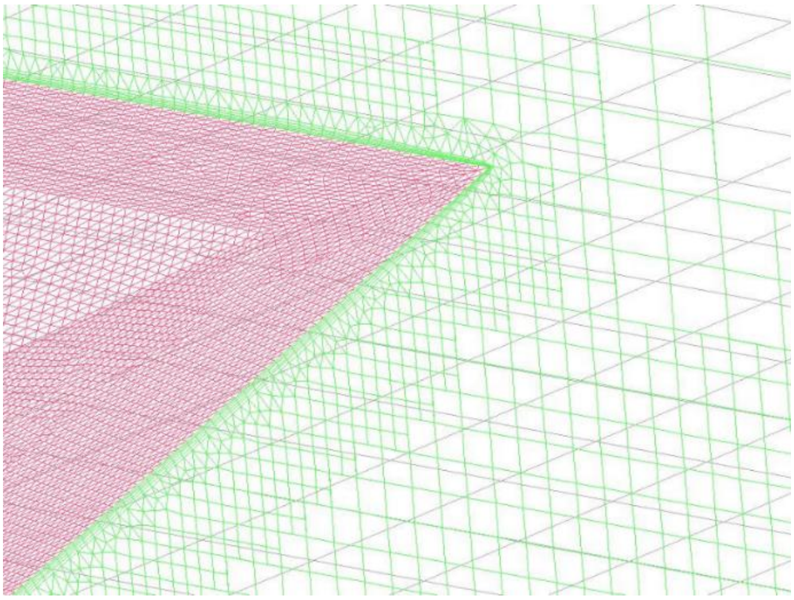

Fig. 1. Structured (right) and unstructured (left) mesh with boundary layer and intermediate zone connected to the refined structured mesh

The most serious limitation in CFD analysis is the number of mesh elements and nodes. In each iteration of the calculation, the hydrodynamic state of the elements is evaluated individually, and their excessive number can massively increase computational complexity and machine time. Hence, it is necessary to keep the number of elements as low as possible, however, not to the detriment of the accuracy of the calculation.

We call a quality mesh when the elements have the same size, are geometrically regular and their distribution is also regular in the discretised area. A suitable choice of element size ensures that the hydrodynamic properties of the flow are captured; however, velocities are decisive for dimensions.

In most cases, due to the limited number of elements, it is not possible to capture the effects in different areas of a domain while meeting all mesh quality criteria. This problem is solved by local refinement of the mesh in computationally demanding and geometrically complex parts of the model. Typical examples are the boundary layer region, leading and trailing edges, free fluid surfaces, etc.

In terms of the motion state of the investigated solids at the boundaries of the computational domain, individual areas of the mesh can be stationary, deformable and dynamic. Generally, dynamic meshes can be used wherever the domain shape changes over time due to the rotational or translational movement of its boundary surfaces. During the simulation, the dynamic mesh and the surrounding mesh zone must be continuously smoothed and remeshed, placing increased demands on computing power [6,10].

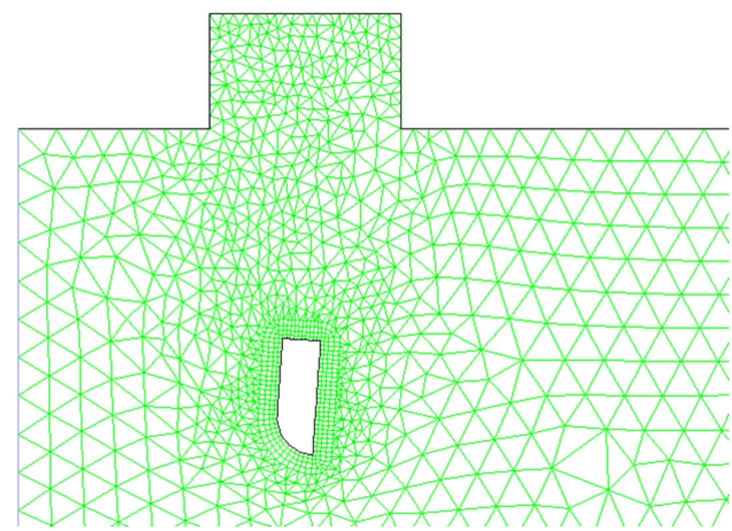

Fig. 2. Dynamic mesh zone around a falling body in a 2D CFD simulation (ANSYS Fluent 17.2 Documentation)

2.3. Methods for solving partial differential equations (PDE) of CFD

Many numerical methods have been developed to address particular physical problems. Their application depends both on the suitability of the method for solving the issue and on the history of development.

By replacing the geometry of the examined area with a mesh of generated nodal points, the flow calculation domain is discretised, thus, allowing the flow equations to be converted into algebraic equations.

Solutions of differential and integral flow equations by discretisation are carried out through various methods, of which the following are the most common:

Finite Volume Method (FVM) - This method in a discretised form retains very reliably, the principles of conservation laws of balanced physical quantities in the control volume and is, therefore, the most widely used CFD simulation apparatus for solving the Navier-Stokes equations.

Finite Difference Method (FDM) - This method is based on the conservative differential form of determining equations. It is a traditional and proven method for numerical solution of partial differential equations. The principle of this method is to transform derivatives into differences in the mesh nodes of the flow field.

Finite Element Method (FEM) - This method uses elements instead of control volumes. Balancing laws are applied to the elements to determine the quantities of the flow field at the nodes of the elements. Unlike the previous two methods, this method also uses interpolation structures to ensure the interdependence of nodal points.

For the solution of algebraic equations, an optimal algorithm (scheme) of the solution is provided, which is the basis for computer software development. Schemes specify iterative processes and may be explicit or implicit. The practical solution is performed using a software and the process should gradually converge to an exact solution. The aim is to achieve a minimum deviation from the exact result.

The best-known solution schemes applied in the CFD area are Euler FTFS scheme, Euler FTCS scheme, Euler FTBS scheme, Upwind scheme, Lax-Friedrichs scheme, Lax-Wendroff scheme, MacCormack scheme, Runge-Kutta scheme. [18].

2.4. Models for turbulent fluid flow

Numerical modelling of turbulent flows is still in the process of research and development, supported by the latest knowledge of mathematics, physics and technical computational methods. However, there is no universal model of turbulence that is generally and effectively applied in all cases. To choose the most suitable model for a particular calculation case, it is necessary to consider the possibilities and limitations of individual numerical models. Turbulent swirls are characterised by length and speed scales and the following methods are appropriate for different scales:

Direct Numerical Simulation (DNS) - This method is suitable for direct solution of a wide range of turbulent vortex sizes, based on Navier-Stokes equations. It does not model swirls but captures turbulence by solving equations with high precision, which requires a very fine mesh. Direct numerical simulation provides a perfect mapping of physical phenomena in a flowing fluid, and its results are considered equivalent to those of experiments. Many mesh elements and a time-dependent process with very small steps lead to the technical unrealisability of simulations in engineering practice.

Large Eddy Simulation (LES) - This method solves only large-scale swirls that can be captured by a coarse mesh at larger time steps. For small turbulent swirls, subgrid models are created and removed by filtration of the turbulent field. Their right combination allows creating a coarse mesh even in engineering tasks whose solution is already realistic with today's computer technology. However, a major disadvantage of the large vortex method is the need to refine the mesh along the body walls in three directions. Various modified methods as well as a RANS/LES hybrid model have been developed to overcome this drawback.

Time-Averaging Method (RANS - Reynolds Averaged Navier-Stokes Equations) - This method has relatively low computational capacity requirements and provides acceptable accuracy. It is being extensively used in engineering simulations. Further, it consists of parametric modelling of turbulent flow by time-averaged values of physical quantities using Reynolds method. Several different RANS methods have been developed for various specific task types, which simplify the modelling of swirls using added transport equations. There is also a method known as DES (Detached Eddy Simulation) which is a transition between RANS and LES. It combines the advantages of both the LES and the RANS numerical modelling methods.

Some of the time-averaging methods are based on Reynolds stresses (RSM), others are based on the Boussinesque hypothesis of turbulent viscosity (for example, k - ε and k - ω). Results calculated by RANS should be confronted with published results or validated by experiment. The most common models of turbulence based on Reynolds averaging and Boussinesque principle, which are also implemented by ANSYS Fluent program are:Spalart-Allmaras Model, k - ε Models, k - ω Models, k - kl - ω Transition Model, Transition SST Model, Reynolds Stress Model (RSM), Large Eddy Simulation (LES) Models, Wall Adapting Local by Eddy Viscosity (WALE) Model, Detached Eddy Simulation, Smagorinsky Lilly Models, Dynamic Kinetic Energy Subgrid Scale Model, Scale-Adaptive Simulation (SAS) Model.

The appropriate optimisation method must be chosen and the specific optimisation task will be formulated only after the precise determination of the maximised and minimised variables (ship parameters) and limiting conditions (waterway restrictions, economic objectives, etc.). A volume of results of technical analyses and the way of their processing also have a major impact [14].

3. RESULTS AND DISCUSSION

Creation of a three-dimensional CFD domain for the analysis of navigational characteristics of vessels is usually started in some solid or surface modelling program.

A solid block with a cavity corresponding to the negative shape of the hull is created. The flow-around is being investigated; this is a typical problem of external flow in CFD. Only one half of the space is modelled because in these cases the boundary condition of symmetry can be applied in the ship's centre plane (XZ). Such a typical 3D model is shown in Figure 3, this particular domain was created for a pontoon-type single-hull test.

Thereafter, after importing the 3D model into the CFD system, the three-dimensional domain body is replaced with a suitable task-specific computational mesh. The mesh is refined in critical areas, special elements are created in the boundary layer and the transition areas. Figure 4 is an overall view of the computational grid, showing that the grid is refined around the body to be flown around and in the free surface area to better capture the physical effects such as turbulence, streamline separation and wave motion.

A cross-sectional view of Figure 5 provides a more detailed insight into the internal structure of such a CFD mesh. It is possible to distinguish individual special types of elements - prismatic in the boundary layer, pyramidal in the transition area and structured surroundings formed by hexahedral elements of different sizes.

After successfully creating the required quality mesh, the computational domain is configured based on input parameters and specifically for the type of analysis. When the system is properly configured, the transient analysis is started, only then the convergence of the calculation is monitored until termination.

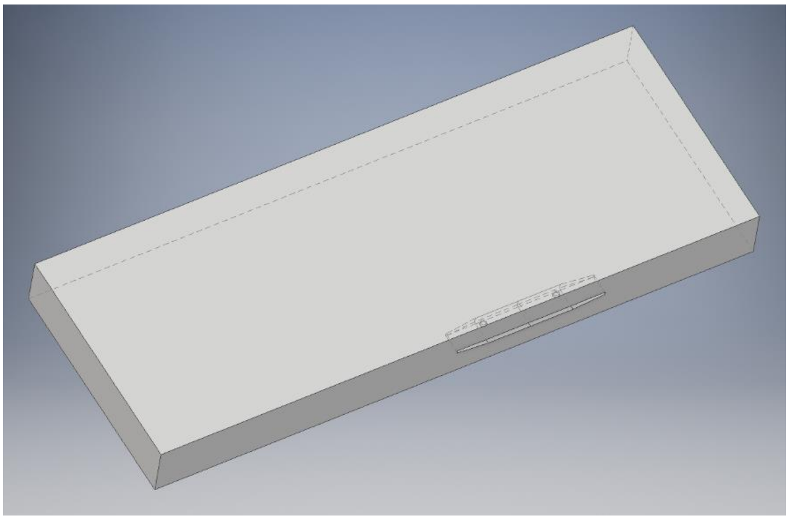

Fig. 3. Example of initial 3D model of a CFD domain intended for vessel flow-around simulation

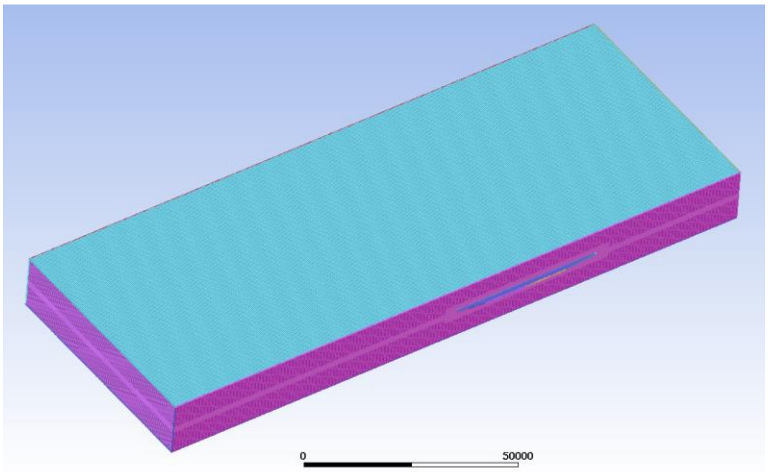

Fig. 4. Computational mesh of CFD domain with refined zones

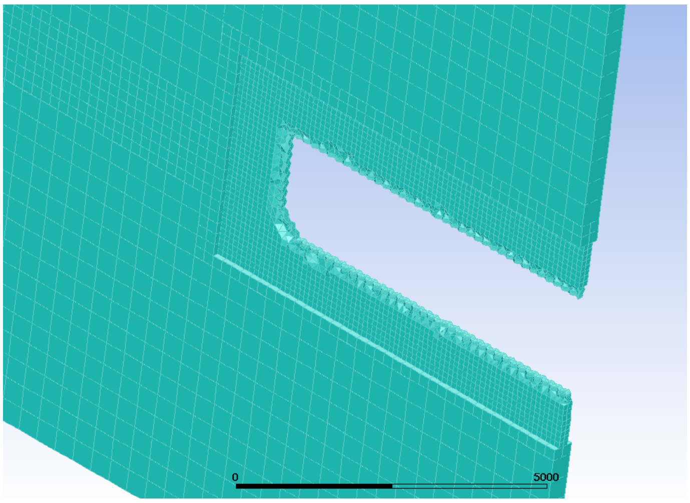

Fig. 5. Internal structure of 3D mesh with different zones

Figure 6 shows the resulting wave field of the vessel. It is an example of CFD analysis of the hull where the physical properties of the flow-around were investigated. In addition, the resistance of the hull was determined at certain constant cruise speeds. It is very important to compare these analysis results with proven technical indicators of other vessels that have been previously model-tested. In most cases, this is an adequate result validation method. By gradual calibration of the CFD task configuration, such as dimensions and proportions of the computational domain, mesh parameters, boundary layer, boundary conditions, initialisation parameters, computational model and numerical methods, it is possible to refine the result and achieve accuracy with a deviation of ± 5%. Such deviations from the results of other sources are satisfactory and the accuracy of CFD analysis is sufficient to perform optimisation tasks.

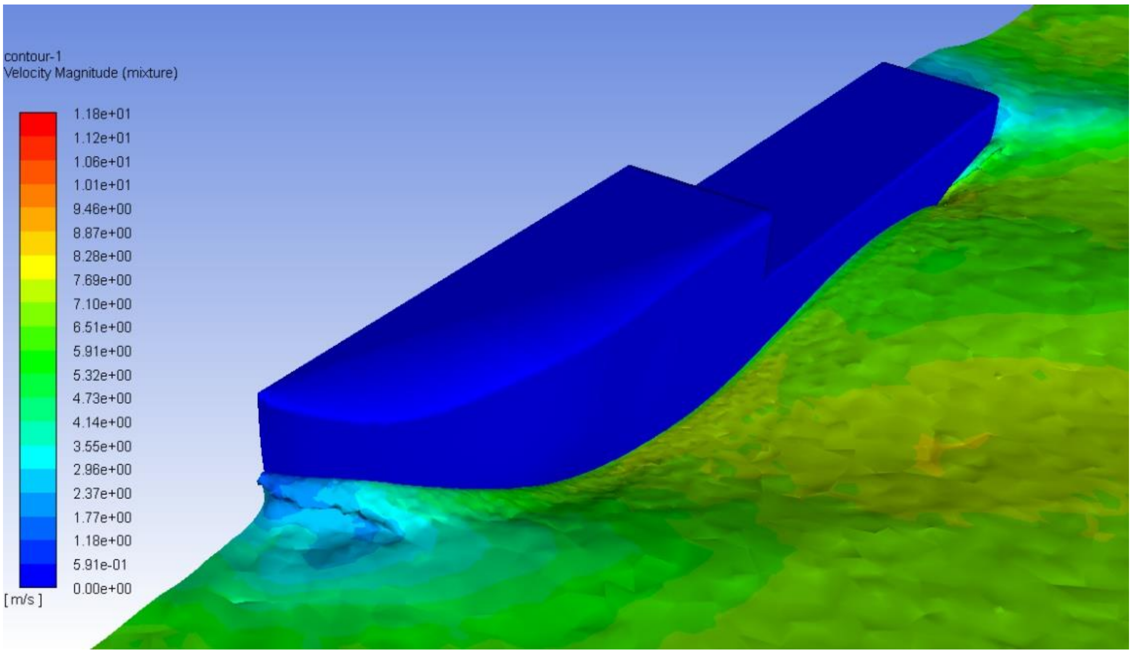

Fig. 6. Example output from CFD analysis of flow-around a vessel – wave pattern

The emergence of the question, what is new in the mentioned methodology, comes to bear at the end of this study. Similar progressive approaches have been applied for several years in various companies and institutions specialised in hydrodynamic development and research.

Here, however, it is a question of finding new ways, for example, researching new principles of hull interaction with multiple propulsion units, determining their optimal number, thrust and location. Of course, such innovation requires major changes in the shape of the hull and in the hydrodynamic parameters of the flow-around. Due to the distribution of propulsion power to several propulsion units over the entire length of the hull and the low draft, the introduction of new types of special propulsors should be considered as well. It is possible to analyse their properties as a single unit submerged in a basin by CFD analysis.

More important is the simulation of their interaction with the hull, which is a much more complex problem due to their increased number and the special hydrodynamically shaped underwater part of the hull. The classic single-propeller and twin-propeller arrangement is presently well documented, and the velocity field influenced by the hull flow-around in the plane of the propeller can be illustrated by the standard wake field diagram. However, current science still has debts in the complex theory of distributed propulsion and their interaction with the hull for cases where some units have to work in the flow field of others, that is, their velocity fields interact.

This is a rather complex issue that the R&D has been trying to resolve in the past by model tests when full functional propulsion units have been mounted on models, scaled in proportions given by the theory of similarity. However, these tests are very time-consuming and costly; therefore, are not appropriate at the research stage or for extensive optimisation processes. Therefore, in recent years, such CFD simulations have already begun, which have already considered the effect of the propulsors on the flow-around and the total flow field nearby the hull. Manufacturers of commercial CFD software have developed tools that allow defining constant flow velocities at several selected locations of the computational domain, and in this way, simulate the operation of the propulsors in a simplified way.

Rotating dynamic meshes that can represent the body shape of the propulsor and its surroundings are available for better accurate analysis. This means that within one domain we can have 3D models of the hull and all propulsors, combining static and dynamic meshes. Of course, this method is computationally demanding, however, it is already practically applicable with the current computer technology.

4. CONCLUSION

The methodological process is designed to solve complex shipbuilding tasks and partial optimisation tasks. Among the basic tasks are, for example, the design and analysis of the characteristics of the new shapes of the ship's displacement hulls, considering the long-term proven hulls of inland waterway vessels.

Another area of application of the methodology is the sizing of the new arrangements on the session: hull - propulsion, considering long-established inland navigation systems. Application of the results can similarly be practised in CFD analysis and comparison of performance parameters of new conceptual and existing solutions.

This methodology is designed to achieve other specific solutions:

- the development of an innovative concept for a new generation of inland waterway vessels,
- simple identification of optimal main ship parameters based on input data, such as navigational restrictions and economic targets,
- support tools or working aids for ship planners, such as the implementation of the proposed methodology,
- preliminary input for economic calculations of shipping establishment and operating costs.

Due to the lower overall complexity, the CFD method has wide application, which in this particular investigation, is the only real alternative for optimisation of distributed propulsion. This is just one of the possible directions for further development of low-draft ships. The ultimate goal is to put an entirely new, innovative type of inland vessel into service in the foreseeable future.

References

1. 90^{th} Session of the Danube Commission. 2018. *Market observation for Danube navigation - results in 2017*. Budapest. Danube Commission.
2. Aguiar F-C., J. Bentz, J-M.N. Silva, A-L. Fonseca, R. Swart, F-D. Santos, G. Penha-Lopes. 2018. “Adaptation to climate change at local level in Europe: An overview”. *Environmental Science & Policy* 86: 38-63. ISSN: 1462-9011. DOI: https://doi.org/10.1016/j.envsci.2018.04.010.

3. Beuthe M., B. Jourquin, N. Urbain, I. Lingemann, B. Ubbels. 2014. "Climate change impacts on transport on the Rhine and Danube: A multimodal approach". *Transportation Research Part D: Transport and Environment* 27: 6-11. DOI: https://doi.org/10.1016/j.trd.2013.11.002.
4. David A., E. Madudova. 2019. "The Danube river and its importance on the Danube countries in cargo transport". *13th International Scientific Conference on Sustainable, Modern and Safe Transport, "TRANSCOM 2019"* 40: 1010-1016. 29-31 May 2019. Novy Smokovec, Slovakia.
5. Doll P., B. Jimenez-Cisneros, T. Oki, N.W. Arnell, G. Benito, J.G. Cogley, T. Jiang, Z.W. Kundzewicz, S. Mwakalila, A. Nishijima. 2014. „Integrating risks of climate change into water management". *Hydrol. Sci. J.* 60: 4-13. DOI: 0.1080/02626667.2014.967250.
6. Douglas J.F., J.M. Gasiorek, J.A. Swaffield, L.B. Jack. 2005. *Fluid mechanics*. Fifth edition, Pearson Education Limited, Harlow. ISBN: 978-0-13-129293-2.
7. Esmailian E., H. Ghassemi, H. Zakerdoost. 2017. "Systematic probabilistic design methodology for simultaneously optimizing the ship hull - propeller system". *International Journal of Naval Architecture and Ocean Engineering* 9(3): 246-255. ISSN: 2092-6782. DOI: https://doi.org/10.1016/j.ijnaoe.2016.06.007.
8. Ferreiro L.D. 1992. "The effects of confined water operations on ship performance: a guide for the perplexed". *Nav. Eng. J.* 104: 69-83.
9. Galierikova A., J. Sosedova. 2018. "Intermodal Transportation of Dangerous Goods". *Nase More* 65(3): 8-11. ISSN: 0469-6255. DOI: 10.17818/NM/2018/3.8.
10. Ganco M. 1983. *Fluid mechanics*. Bratislava: ALFA Bratislava. ISBN: 63-745-83.
11. Habersack H., T. Hein, A. Stanica, I. Liska, R. Mair, E. Jager, Ch. Hauer, Ch. Bradley. 2016. "Challenges of river basin management: Current status of, and prospects for, the River Danube from a river engineering perspective". *Science of The Total Environment* 543(A): 828-845. DOI: https://doi.org/10.1016/j.scitotenv.2015.10.123.
12. Harvald S.A. 1977. "Wake and thrust deduction at extreme propeller loadings for a ship running in shallow water." *RINA Suppl. Pap.* 119.
13. Kim D.H., J.K. Paik. 2017. "Ultimate limit state-based multi-objective optimum design technology for hull structural scantlings of merchant cargo ships". *Ocean Engineering* 129: 318-334. ISSN: 0029-8018. DOI: https://doi.org/10.1016/j.oceaneng.2016.11.033.
14. Kudelas D. 2017. *Basics of computer flow modelling and visualization*. Kosice: Faculty BERG TU, Kosice.
15. Lackenby H. 1963. "The effect of shallow water on ship speed". *Shipbuild. Mar. Eng.* 70: 446-450.
16. Lobanova A., S. Liersch, J.P. Nunes, I. Didovets, J. Stagl, S. Huang, H. Koch, M.R.R. Lopez, C.F. Maule, F. Hattermann, V. Krysanova. 2018. "Hydrological impacts of moderate and high-end climate change across European river basins". *Journal of Hydrology: Regional Studies* 18: 15-30. ISSN: 2214-5818. DOI: https://doi.org/10.1016/j.ejrh.2018.05.003.
17. Macháčková A., P. Kuchta, Z. Klečková, R. Kocich, J. Szwed. 2016. "Numerical simulation of the heat treatment of the weld for steam generator". *Metalurgija* 55(4): 741-744.
18. Molnar V. 2011. *Computational Fluid Dynamics - Interdisciplinary Approach with CFD*. Bratislava: STU Bratislava. ISBN: 978-80-8106-048-9.

19. Nouasse H., A. Doniec, G. Lozenguez, E. Duviella, P. Chiron, B. Archimede, K. Chuquet. 2016. "Constraint satisfaction problem based on flow graph to study the resilience of inland navigation networks in a climate change context". *IFAC-PapersOnLine* 49: 331-336. DOI: https://doi.org/10.1016/j.ifacol.2016.07.626.
20. Raven H. 2012. "A computational study of shallow-water effects on ship viscous resistance". *Proceedings of the 29th Symposium on Naval Hydrodynamics*. Gothenburg.
21. Raven, H. 2016. "A new correction procedure for shallow-water effects in ship speed trials". *Proceedings of the 2016 PRADS Conference, Copenhagen.*
22. Rotteveeel E., R. Hekkenberg, A. Ploeg. 2017. "Inland ship stern optimization in shallow water". *Ocean Engineering* 141: 555-569. ISSN: 0029-8018.
23. Rotteveel E., R. Hekkenberg. 2015. "The influence of shallow water and hull form variations on inland ship resistance". *Proceedings of the 12th International Marine Design Conference.* "*IMDC 2015*". 11-14 May 2015. Tokyo, Japan.
24. Saha G.K., K. Suzuki, H. Kai. 2004. "Hydrodynamic optimization of ship hull forms in shallow water". *J. Mar. Sci. Technol* 9: 51-62. DOI: 10.1007/s00773-003-0173-3.
25. Schlichting O. 1934. *Ship's resistance to limited water depth: Resistance of ships to shallow water.* Jahrb. der Schiffbautech 35: 127.
26. Sun H., O.M. Faltinsen. 2012. "Hydrodynamic forces on a semi-displacement ship at high speed". *Applied Ocean Research* 34: 68-77. ISSN: 0141-1187. DOI: https://doi.org/10.1016/j.apor.2011.10.001.
27. Tuck E. 1978. "Hydrodynamic problems of ships in restricted waters". *Annu. Rev. Fluid Mech* 10: 33-46. DOI: 10.1146/annurev.fl.10.010178.000341.
28. Wang X.M., X.L. Wu, W.Q. Zhou. 2019. "Analysis of oxygen enriched combustion characteristic of 350 MW utility boiler based on computational fluid dynamics". *Metalurgija* 58(3-4): 223-227.
29. Zhao L-e. 1984. "Optimal ship forms for minimum total resistance in shallow water". *Schr. Schiffbau.* DOI: 0.15480/882.930.

Received 04.07.2020; accepted in revised form 19.10.2020