Towards Green Marine Technology and Transport – Guedes Soares, Dejhalla & Pavletić (Eds)
© 2015 Taylor & Francis Group, London, ISBN 978-1-138-02887-6

Ship hull form design and optimization based on CFD

A.A. Aksenov
Capvidia/TESIS, Moscow, Russia

A.V. Pechenyuk
Digital Marine Technology, Odesa, Ukraine

D. Vučinić
Vrije Universiteit Brussel, Brussels, Belgium

ABSTRACT: Nowadays, the Computational Fluid Dynamics (CFD) represents a standard design practice in solving the ship hydrodynamics problems, worldwide. In recent years CFD is actively in used by numerous maritime design organizations and related educational institutions. CFD is applied for propulsion research of the new-built or modernized ships, as it is well suited for the practice in the design and optimization of the ship hull form. The presented results are also including the verification/validation analysis of the KCS hull form from the Gothenberg-2000 workshop has been developed by the authors of this paper, and through this comparative study, the best practices learned in this process, are shown. In this paper the FlowVision code is used as the CFD tool, integrating a new method of in-detail hull form design based on the wave-based optimization. The hull form design and its optimization are based on the systematic variation of the longitudinal distribution of the hull volume, while the vertical volume distribution is fixed or highly controlled. Such design process is underpinned with the respective data analysis of the obtained results, which are presented as the optimum distribution of the required hull volume. The final result is the optimized designed hull form, which shows interesting characteristics, as its resistance has decrease by 8.9% in respect to the well-known KCS hull form.

1 INTRODUCTION

Hydrodynamic problems of ship design are well known to be complex and exacting. A conventional approach to solve them is comparatively expensive and long-term experiment in the towing tanks or other laboratory facilities. Since 1985 the numerical methods in Computational Fluid Dynamics (CFD) are used in ship design, first of all, for the purposes of the hull form optimization and also for the optimize/design of other components in the ship propulsion system. The theoretical background of CFD came into sight already in the 1960s, however its implementation in industry became only possible due to the rapid progress of the computer technology.

As it is well known, the panel methods based on the potential non-viscous flow model have used singularities as the first computational model effectively applied to the ship design. They had a lot of drawbacks due to the neglecting or incomplete modeling of viscosity. Nevertheless, the panel methods provided quite realistic qualitative results of the hull flow behavior, including a wave pattern, which complexity has been leveraged to the available computing resources at that time.

While the computing capabilities were rising, another CFD approach based on numerical solving the differential equations of viscid flow (Navier-Stokes equations) was developed. In general, these equations may be solved by the numerical Finite Element (FE) or Finite Volume (FV) methods, but direct approach puts heavy demands on the resources needed for the practical projects. Therefore, special numerical methods based on the Navier-Stokes equations were developed to simplify the problem, with the objective to decrease the computational effort. At the present time, a method of solving the Reynolds-Averaged Navier-Stokes equations (RANS) is the most popular one used in the practical ship hydrodynamics. This promising approach provides flexible and exact simulating conditions, which made them very popular since 2000s. An example of more detailed historical overview can be found in the book (Larsson & Raven 2010).

According to the materials of the Gothenburg 2010 Workshop on Numerical Hydrodynamics (Larsson et al. 2011), there are about 20 software packages, which allow high precision modeling of the viscid hull flow with a free surface. Some of them are well-known commercial products;

the other ones are developed by educational or research organizations. All these codes solve Navier-Stokes equations by numerical methods; the majority of them use RANS approach for the hull flow simulation. Theoretical grounds of the leading codes are very similar, but there is a huge variation in supporting models, grid technologies and programming features. The validation of the results showed that the mean accuracy in the resistance estimation of the state-of-the-art CFD methods is still slightly below the experimental level, but already is comparable to it. The FlowVision CFD code is used for getting simulation results in this paper. Detailed and comprehensive information about its CFD capabilities for ship hydrodynamics, including technical features, experience of use in ship design, verification and validation results, as well as the data from up-to-date scientific study, is gathered in the paper.

2 CAE-CFD TOOLS FOR SHIP DESIGN

2.1 *Technical features*

FlowVision is based on the Finite Volume (FV) method, which is the industry standard in modern CFD. It is able to simulate both compressible and incompressible flows of liquid and gas, has five turbulence models, including low-Reynolds turbulence models.

3D Partial Differential Equations (PDE) describing different flows, viz., the mass, momentum (Navier-Stokes), and energy conservation equations are integrated. If the flow is coupled with additional processes like turbulence, free surface evolution, combustion, etc., the corresponding PDEs are added to the basic equations. All together the PDEs, state equations, and closure correlations (e.g. wall functions) constitute the mathematical model of the flow. Implicit velocity-pressure split algorithm is used for integration of the Navier-Stokes equations by FV method.

The main feature of FlowVision versus other codes is the automatic generation of computational grid. Up to 90 percent of the user's working time is usually occupied by generation of the computational grid while preparing FV or FE simulations. In FlowVision this problem was resolved by using the subgrid geometry resolution method for generation of the Cartesian Adaptive Locally Refined Grid (CALRG). The essence of this method (Aksenov et al. 1998) is a Boolean subtraction of the volume, which is determined by closed surface and borders the computational domain, from the Cartesian computational grid. In fact the cells of the computational grid, when are crossed by the freeform surface of the computational domain, are converted into complex polyhedrals with an approximation of the solved equations inside them by high-order schemes. In areas near the boundaries with high gradients of the flow parameters, an additional grid resolution is carried out by dynamic adaptation of the computational grid (given contiguous cells are divided into smaller ones). As a result, a user forms general configuration of the initial Cartesian grid only, while the computational cells with complex geometry near boundary conditions are meshed automatically. This unique approach to grid generation provides a natural link with CAD geometry and FV mesh.

URANS approach is used in the simulations of a turbulent hull flow. It is based on solving the time-dependent equations of fluid motion and the two-equation turbulence model of k-ε type. Simulation of the laminar-turbulent transition in the boundary layer is available with the help of special modification 'KOLOKOL' of the k-ε turbulence model. The modified Volume of Fluid (VOF) method is used for a free-surface modeling. It allows the highly accurate simulation of wave pattern on free surface around a ship. The important FlowVision feature for ship hydrodynamics is the possibility of applying simultaneously the moving body modeling with the free surface simulation, as example, is the modeling of the dynamic trim.

FlowVision includes all the necessary capabilities and technologies for the wide-range of applications in the maritime design practice. Not only representative for the towing test simulations, but also a self-propelled simulation of the ship movement, both with actuator disk and the more realistic one, with the propeller model presence can be performed (Vučinić et al. 1984, 1994). In addition, other important ship propulsion, sea keeping and maneuverability models are available for simulations. A very advance use represents the ABAQUS integration through Multi-Physics (MP) Manager support, which enables the very complex Fluid-Structure Interaction (FSI) simulations (e.g., hydro-elastic problem) to be done.

2.2 *Best practice lessons learned*

The DMT specialists have carried more than 50 research projects based on CFD to the new-built or modernized ships. The main activity was treating the hull form design and optimization, and propulsion calculations. An outstanding recent project is described in the paper (Aksenov et al. 2014). The experience shows that the CFD approach has made possible significant costs savings, in terms and flexibility of the working process when compared with the conventional experiments.

In our opinion, the main advantages can be described as follows. 3D virtual ship hull models can be easily and rapidly modified compared to the experimental models. A significant part in the cost of experimental research corresponds to production of models, and thus directly limiting the optimization capabilities. FlowVision allows changing geometry of the researched object even in the process of simulation (without loss of the previous data). This technique saves a lot of time. Furthermore, to decrease the time to solution a user may access to actually "unlimited" High-Performance Computational (HPC) resources as it done during the UberCloud HPC Experiment (Jackson et al. 2013). Access to HPC resources helps to avoid too long time to compute, limiting the usefulness and usability of the CFD approach for complex problems, as it supports very flexible and advanced HPC technologies: the same interface may easily start a solver on multiple cores within a PC or on remote supercomputers.

An extra advantage of the CFD simulation versus the experimental model tests is a large volume of visual information detailing the local flow characteristics, found very important to be available to the designers (Vučinić 2010). The examples of this information are: wave patterns as isolines or isosurface, pressure distribution, streamlines, velocity vectors etc. both on the hull surface and in any region of the flow. These capabilities are very useful for faster search for optimal design solutions.

In the current market, CFD may improve the feasibility of the propulsion research for the low-budget projects of the small vessels, such as river ships, technical ships or yachts. On the other hand, complete projects of the large and expensive vessels need the CFD simulations, as an effective optimization tool. Undoubtedly, all aforementioned advantages of the CFD approach were effectively applied to design practice, because of the continuous and transparent correlation with the reliable experimental data. The CFD methods produce numerous different classes of errors, unlike transparent experimental procedures based on the similarity theory. It is not evident that the result of CFD simulation will always agree with physical reality. Thus careful research of the quality of a method—verification and validation, is necessary to ascertain this approach.

3 VERIFICATION AND VALIDATION RESULTS FOR THE KCS HULL FORM

In the field of naval architecture the most competent recommendations in verification and validation of the CFD methods were developed within an international workshop on the numerical prediction of ship viscous flow. In the workshop Gothenburg-2000 3 modern hull forms with reliable experimental data were introduced to be the validation test cases. The most general case among them is the containership KCS, a ship of moderate specific speed and fullness. The numerical analysis of the KCS hull flow behavior according to the formal procedures of the workshop was carried out with the help of CEA FlowVision developed by the authors of this paper in 2013. Findings were compared with the experimental data and computational data of other participants.

3.1 *The object of research*

The KRISO Container Ship (KCS) is a well-known benchmark case for which measurements have been made in Korea and Japan (Kume et al. 2000). Hull lines of the KCS are shown in Figure 1.

The research was focused on towing test of the bare hull in model scale (cases 2.1 and 2.2).

3.2 *Physical model*

The physical model of incompressible fluid based on RANS and continuity equations was used. Vector RANS equation is:

$$\frac{\partial V}{\partial t} + \nabla(V \otimes V) = \\ = -\frac{\nabla p}{\rho} + \frac{1}{\rho}\nabla\left((\mu + \mu_t)(\nabla V + (\nabla V))^T\right) \quad (1)$$

where V = velocity; t = time; p = pressure; ρ = density; μ = molecular viscosity; μ_t = turbulent viscosity.

A modified VOF method was used for the free surface modeling. This method determines a fill function F with the help of transfer equation (2). In the fully filled cells $F = 1$; in the empty cells $F = 0$. The cells with fractional F values correspond to an interface. Free surface is approximated by isosurface with $F = 0.5$.

$$\frac{\partial F}{\partial t} + V\nabla F = 0 \quad (2)$$

For using RANS approach it's necessary to determine turbulent viscosity μ_t. Various turbulence models can be applied for this purpose. The standard k-ε turbulence model was used in the present study.

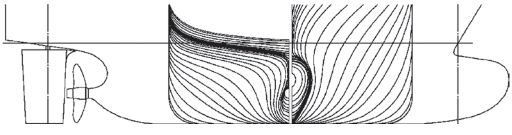

Figure 1. Hull lines of the KCS.

The standard k-ε turbulence model is considered to be reliable and simple when the dimensions of the computational cells near wall, as well as velocities of flow, significantly vary. If there is no significant flow separation, the boundary layer is completely turbulent (artificial turbulence generation by wires in towing tank tests), and for such case the standard k-ε model is found to be quite accurate for the CFD simulations runs.

Test case 2.2 requires modeling the free trim under action of the hydrodynamic force. The free trim was modeled by the moving body feature. The ship form was modeled as the geometrical object, which has 6 Degrees of Freedom (DOF). However, only 2 degrees are used in the simulation of towing test: (1) a movement along vertical axis (sinkage) and (2) a rotation about transverse horizontal axis (trim). An important feature is the ability of rebuilding the cells crossed by the ship body movements, which is done between iterations. Although this feature needs additional computational resources, there is no significant drop in performance thanks to CALRG approach applied for the meshing.

3.3 *Numerical model*

When towing speed is constant, it's reasonable to simulate the inverted flow around a hull (like in water tunnel instead of considering water tank conditions). In this case the computational domain is shorter, and the hull is nearly static in respect to the grid. Taking this into account, a box-shaped Computational Domain (CD) was defined, and the corresponding Boundary Conditions (BC) have been set (Fig. 2). The transverse sizes of CD were chosen larger than the once recommended for the towing tests, in order to avoid the tunnel wall effects. For the same reasons, the free outlet BC was set on the sides and the bottom of CD. Due to the KCS symmetrical hull form, only the respective half of the ship form flow has been simulated.

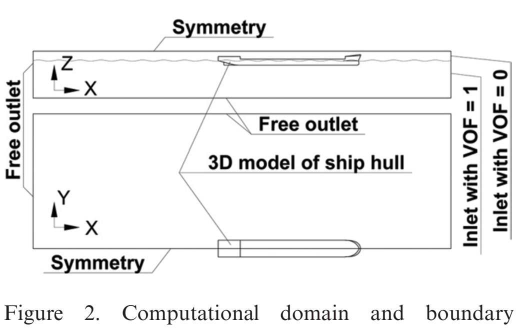

Figure 2. Computational domain and boundary conditions.

BC on the hull surface have been defined as the wall with the logarithmic law for the velocity distribution, which is a usual setup for the wall function in fully developed turbulent boundary layer.

The computational grid was formed on the basis of the initial Cartesian grid, with significant number of non-uniform cells. The small cells were concentrated near the ship ends and near the free surface, by taking into account the wave-generation model applied during the simulations. For some volumes around the hull form, the cells have been reduced in size by 2 (adaptation level 1). The mesh sensitivity study was made to observe the impact of the mesh density on the towing resistance at $Fr = 0.26$. The grid structure was similar, while the cells sizes were varying. This study had shown that mesh density has the quite large effect—see Figure 3, on the obtained results. It must be stated that a fully grid non-depended result was not achieved even when the maximum mesh density was applied. However, there is a clear indication that the simulation error decreases when the grid is more refined. The grid with about 3.7 M computational cells (maximum density) was used for the validation stage. This size responds well to the average level that was shown in the workshop Gothenburg 2010.

In practice, when the grid reaches the size of 3–4 M cells, simulations with a free surface can be carried out efficiently only by the use of HPC resources. The HPC resources of NRC "Kurchatov Institute" were used in this study.

3.4 *Comparison of the results for the fixed trim (cases 2.1 and 2.2a)*

According to the case 2.1 a comparison of the wave pattern data was made. In Figure 4 a comparison of free surface elevation as z/L isolines (global wave pattern) is shown at $Fr = 0.26$, $Re = 1.4E07$. The CFD data correlates to the Experimental Data (EFD) well near the hull, but quite poorly away from it. In Figure 5, a comparison of the free

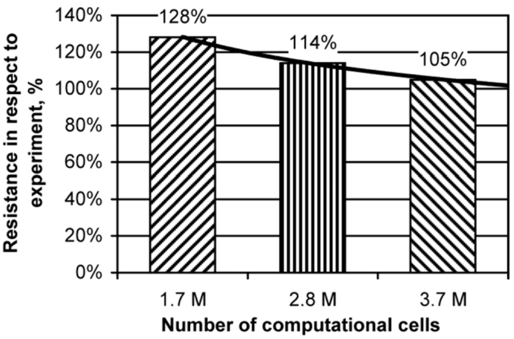

Figure 3. Results of the mesh sensitivity study.

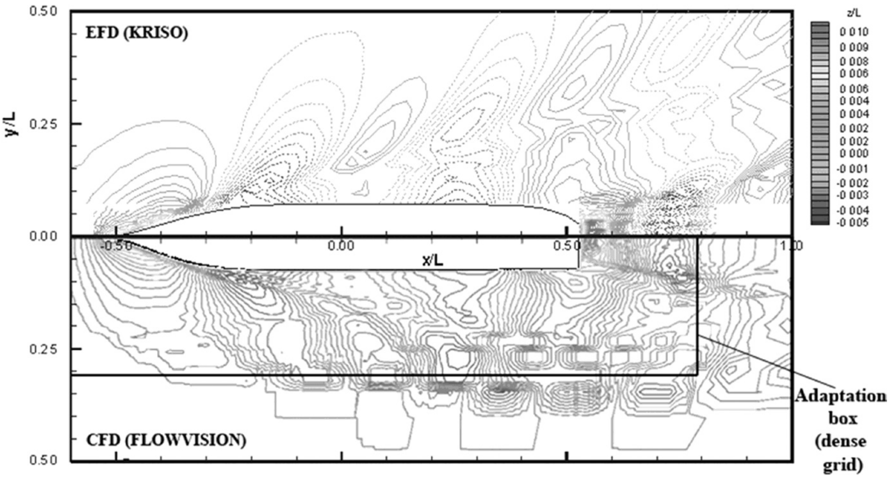

Figure 4. Free surface elevation, global wave pattern.

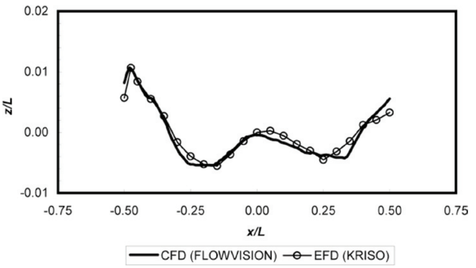

Figure 5. Free surface elevation along the surface of the hull.

surface elevation along the surface of the hull is shown at the same model speed. It can be seen that the CFD data and the EFD data are in good general correlation.

The poor correlation at some distance from the hull can be explained by a hindrance to wave propagation on the borders of the adaptation box (see Fig. 4). Outside the box the cells become larger too fast. In grids without an adaptation in the volume wave patterns are smoother. However, the used grid provides an advantage around the hull. It is very important for the accuracy of towing resistance. A very good correlation may be noted behind the transom, which is usually uneasy to obtain.

A comparison of the resistance coefficients is shown in Table 1.

Table 1. Comparison of the resistance coefficients (case 2.2 a).

EFD (KRISO)		CFD (FlowVision)		Error in C_T
$C_R \times 10^3$	$C_T \times 10^3$	$C_R \times 10^3$	$C_T \times 10^3$	E
0.7294	3.5600	0.7549	3.5855	+0.7%

Table 2. Comparison of the resistance coefficients (case 2.2 b).

Speed	EFD (KRISO)		CFD (FlowVision)		Error in C_T
Fr	$C_R \times 10^3$	$C_T \times 10^3$	$C_R \times 10^3$	$C_T \times 10^3$	E
0.1733	0.4650	3.4997	0.6169	3.6517	+4.3%
0.1949	0.4500	3.4232	0.7501	3.7233	+8.8%
0.2166	0.4800	3.3998	0.6744	3.5942	+5.7%
0.2382	0.5780	3.4506	0.8214	3.6940	+7.1%
0.2599	0.8320	3.6626	0.9109	3.7415	+2.2%
0.2707	1.1460	3.9572	1.2905	4.1017	+3.7%
0.2816	1.6600	4.4528	1.5880	4.3807	−1.6%
			Mean error		+4.3%

3.5 *Comparison of the results for the free trim (case 2.2b)*

A comparison of the resistance coefficients for the free trim is shown in Table 2. The comparison showed good accuracy in resistance when $Fr \geq 0.26$ (mean error is +1.6% only). At slower speed the CFD data demonstrate stable overestimation (mean error +6.5%). This is an interesting result. On the one hand, the experimental dependence of residual resistance coefficient C_R versus Fr shows that the wave resistance is very low at $Fr < 0.26$. This may cause difficult conditions of

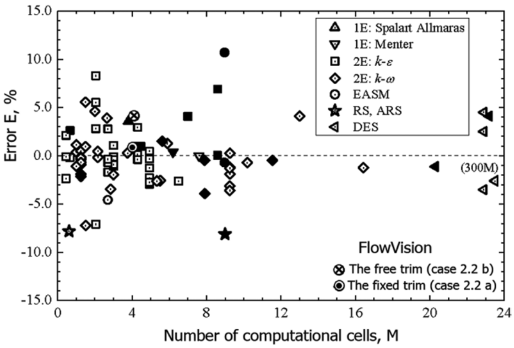

Figure 6. Comparison error versus grid size.

simulation with free surface. On the other hand, the same grid was used for various speeds. As a result, at lower speed the grid was rougher in respect to the waves. The mesh sensitivity study showed that a rougher grid leads to higher resistance in this case.

3.6 *Discussion*

The obtained errors were compared to the errors of the workshop Gothenburg 2010 participants in Figure 6. It can be seen that the CFD errors are of the same level, when the similar grid and turbulence models are used. The difference between errors for the fixed trim and the free trim is mainly due to the influence of the Froude number. The fixed trim case was simulated at $Fr = 0.26$ only. At $Fr \geq 0.26$ errors were also very small for the free trim. But the results were less accurate at the lower Froude numbers. Our analysis showed that the main probable reason is the effect of the grid rather than the effect of trim parameters. A special grid is apparently necessary for the low Froude numbers to provide good resolution of the smaller waves near the hull.

4 OPTIMIZATION OF THE KCS HULL FORM

Optimization of hull lines for the minimum resistance to movement is a problem of current interest in ship hydrodynamics. The deep theoretical researches were undertaken to find the hull form of minimum resistance (Zhukovsky 1937; Havelock 1934; Weinblum 1930, Pavlenko 1956; Sretensky 1957; Sizov 2006; Inui 1962 and many others). Although the important results were obtained, the problem had not been resolved.

In practice, lines design is to some extent an art. The approaches to decrease the ship resistance are based on the model experiment and/or CFD simulation, following the trial and error method. The scientific methods of linear programming or experiment planning can't be involved because of 2 main reasons.

Firstly, the model test is too expensive and long-term, and the CFD simulation requires too many computational resources when numerous variants of the hull form are studying. Secondly, the geometry of the ship hull is a smooth surface. It is not clear yet, which variables describe the hull form in the best way to support the desired optimization process.

4.1 *Approach to modification of a hull form*

In analytical research, the hull form was modeled with the equation of surface. In this case the coordinates of the points can be considered as the optimization variables. However, when both the model tests and/or CFD simulation are made, it's difficult to make optimization by varying the individual ship hull form points, as possible when applying the theoretical models, as the detected resistance variations are too small.

The generalized parameters of the hull form geometry are the well-known elements for the theoretical drawing. However, applying all of them together has too complex effect on resistance. Thus, the existing statistical methods are not suitable for the in-detail optimization of the hull form. In our opinion, the principal dimensions and their ratios can be removed from such consideration, as their influence is already well studied. In addition, there are a lot of requirements, which are not related to hydrodynamics.

Therefore, if some initial hull form is predefined, it can be easily modified by changing the elements defining the longitudinal and vertical distribution of the required volume. These distributions are visually described by sectional curves and their respective areas. It's well known from the theoretical studies that the vertical distribution can't be optimized, thus it can be neglected for the wave resistance optimization. Thereby, only the variation of the longitudinal distribution of the hull volume may be considered in the optimization procedures, while its vertical volume distribution is fixed or highly controlled. From theoretical point of view, this approach to define the optimum ship hull form may be described as an analogue of the variational method.

The variations of the longitudinal distribution of the volume were set on 3 segments of the sectional area curve of the KCS foreship (see Fig. 7). Each modification is a new segment of the curve, which is smoothly connected to the rest. Two volume increments of a curve segment have opposite

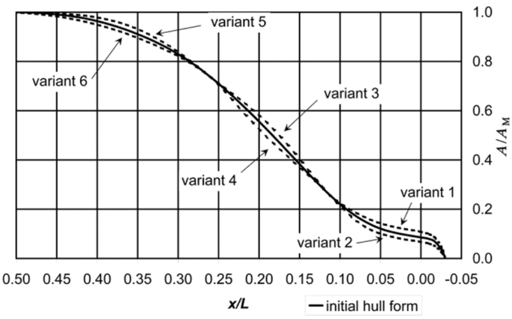

Figure 7. Modifications of the KCS foreship on the sectional area curve.

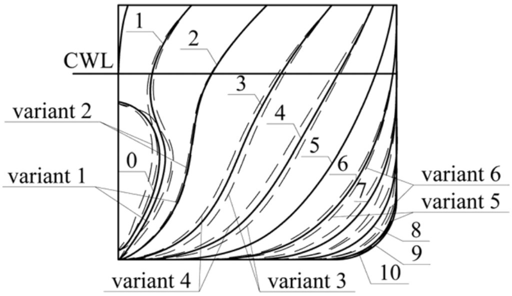

Figure 8. Modifications of the KCS foreship on the lines plan.

signs, but the same value 0.4% in respect to the hull volume.

The modifications can be then transferred from the sectional area curve to the lines plan. The various methods for the frame transformation may be involved. Affine transformation of the frames is the most simple and fixes the vertical volume distribution. However, a more complicated method was used for the KCS foreship to preserve the form of the deck (see Fig. 8). As the result, six variants of the hull forms were obtained and the corresponding 3D models were prepared.

4.2 *Results of CFD simulations*

The CFD towing tank simulations were carried out for the initial hull form and the six variants by the method described in above section 3. Unlike done for the test cases, a full-scale calculations were made at $Fr = 0.28$ with fixed trim.

According to the simulation results, modifications of the KCS foreship correspond to resistance variation in the range 1.3%–6.5% (see Fig. 9). Almost, exclusively the pressure resistance was involved in the variation.

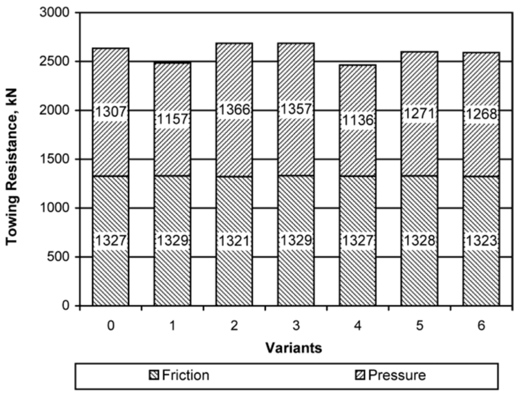

Figure 9. Results of the CFD simulations for the hull form variants.

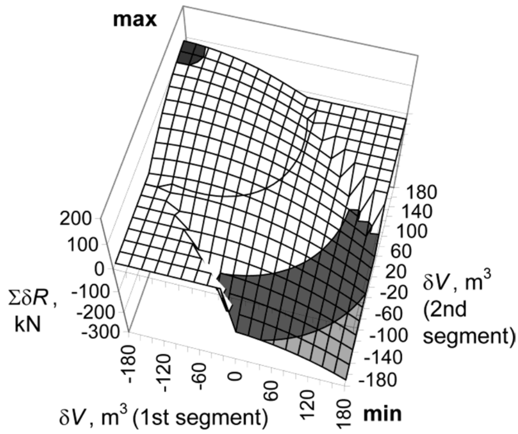

Figure 10. Approximation of the simulation results for searching the optimum.

4.3 *Formation of the optimum hull form and its analysis*

Formation of the optimum hull form is based on an approximation function $\delta R(\delta V)$ for the dependence between the increments of resistance and the increments of volume. This function can be reconstructed for the constant volume condition if the effects of the volume increments are considered as independent from each other. A. Pechenyuk formulated the corresponding hypothesis in his PhD thesis. Its main results are published in the paper (Pechenyuk 2014).

When number of the modified segments is 3, the function $\delta R(\delta V)$ can be shown as 3D surface (see Fig. 10). The volume increments of the first and the second segments (starting from the fore) are plotted along the axes of arguments, while

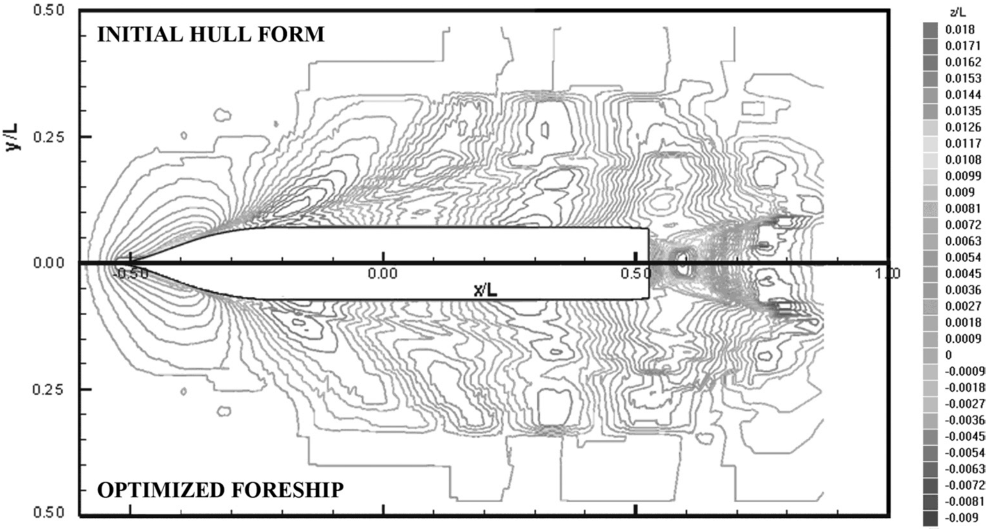

Figure 11. Free surface elevation, global wave pattern of the initial and optimized hull at *Fr* = 0.28.

resistance increments are plotted along the axis of function. The volume increment of the third segment becomes a depended variable when the hull volume is constant.

The minimum resistance corresponds to transferring the volume from the second segment to the first. The maximum effect (within the considered range of modifications) was estimated as 319.8 kN or 12.1%. Unfortunately, the aforementioned hypothesis doesn't take into account the viscid phenomena, and prediction can't be completely accurate. CFD simulation of the 3D model with optimized hull form showed the decrease in resistance 243.4 kN or 8.9%.

Visualization of the wave patterns (Fig. 11) shows the decrease of resistance. It is very difficult to see the decrease of resistance from the figure, as we are able to see only the changes of the waves pattern, which is accompanied by an attenuation of the transversal wave components, and the intensification of the diverging wave components. This result matches the well-known physical law described by T. Havelock (Havelock, 1934).

5 CONCLUSION

The design and optimization of the ship form is presented by the application of the CFD based approach which data has been simulated by the FlowVision software.

In detail description of the process has been presented in showing a step-by-step procedure based on the analog to the variational method.

A significant decrease of the ship resistance for the analyzed models has been achieved and validated by the comparison with the well-known KCS database.

The final result is the optimized designed hull form, which shows interesting characteristics, as its resistance has decrease by 8.9% in respect to the KCS hull form.

REFERENCES

Aksenov A., Dyadkin A. & Pokhilko V. 1998 Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method. *Proc. 1998 ASME Pressure Vessels and Piping Division Conference* Vol. 377-1., San Diego (California): ASME PVP.

Aksenov A.A., Zhluktov S.V., Petrov A.S., Pechenyuk A.V., Stankov B.N. & Yu. Cloud 2014 Design of Hull Lines Using Modern CFD Software Package FlowVision. *Proc. of 3rd National CFD Conf. of Shipbuilding and Marine Engineering* (pp. 262–267). Dalian: Dalian University of Technology.

Havelock T.H. 1934 Wave Patterns and Wave Resistance. *Trans. of the Inst. of Naval Architects* 76: 430–446.

Inui T. 1962. Wave-Making Resistance of Ships. *Trans. SNAME* V. 70: 283–326.

Jackson A., Lorenzana J., Pechenyuk A. & Aksenov A. 2013 CAE Simulation of Water Flow Around a Ship Hull. *The UberCloud HPC Experiment: Compendium of Case Studies*.

Kume K., Ukon Y. & Takeshi H. 2000 Measurements of Surface Pressure and Local Velocity Field around a KCS Model and Uncertainty Analysis. *Proc. of the Gothenburg 2000 Workshop*. Gothenburg: Chalmers University.

Larsson L. & Raven H.C. 2010. *Ship Resistance and Flow.* Jersey City (USA): SNAME.

Larsson L., Stern F. & Visonneau M. 2011 CFD in Ship Hydrodynamics—Results of the Gothenburg 2010 Workshop. *Proc. of the IV International Conference on Computational Methods in Marine Engineering* (pp. 17–36). Barcelona: CIMNE.

Pavlenko G.E. 1956 *Water Resistance to Ship Movement*. Moscow: Maritime Transport (in Russian).

Pechenyuk A. 2014 About Increasing in Efficiency of a Propulsion Complex by Means of Bow Form Improvement. *Ship Power Plants: Scientific and Technical Digest* 34: 87–97. Odesa: ONMA (in Russian).

Sizov V.G. 2006 About a Method of Ship Form Improvement. *ONMU Herald* 19: 14–19. Odesa: ONMU (in Russian).

Sretensky L.N. 1957 Sur la resistance due aux vagues d'un fluid visqueux. In *Proc. Symp. on the behavior of ship in a seaway*, Wageningen 2: 729–733.

Vučinić D. 2010 *Development of a Scientific Visualization System: CFView—Computational Field Visualization System and Object-Oriented Software Methodology*. LAP Lambert Academic Publishing, ISBN 978-3-8383-3500-1, 280 pages.

Vučinić A., Hirsch C., Vučinić D., Dener C. & Dejhalla R. 1994 Blade Geometry and Pressure Distribution Visualization by CFView Method. *Strojarstvo* vol. 36, No. 1–2: 45–48.

Vučinić A. & Vučinić D. 1984 Kv-Series of Screw in Nozzles—regression analysis of open water test results. *Proc. of the* 6th *symposium "Theory & Practice in Shipbuilding—in memoriam Prof. L. Sorta",* Beograd:

Weinblum G. 1930 Anwendung der Michellschen Widerstandstheorie. *Jahrbuch der Schiffbautechnischen Gesellschaft.* Berlin: Springer Berlin Heidelberg.

Zhukovsky N.E. 1937 About the Wake Wave. *Complete set of works, Vol. 4.* Moscow: ONTI NKTP USSR (in Russian).