Optimization of the research vessel hull form by using numerical simulation

Jonas Čerka, Rima Mickevičienė, Žydrūnas Ašmontas, Lukas Norkevičius, Tomas Žapnickas, Vasilij Djačkov, Peilin Zhou

ABSTRACT
This paper presents results of hull form optimization of a research vessel using numerical simulation. The object of the study is a newly built multi-purpose catamaran type research vessel, meant for geological and biological researches. The vessel reaches its full speed by using only 70% of total engine power. Further increase in power leads to hardly noticeable increase in speed. Vibration appears additionally in the wheelhouse area. This leads to study possible causes of negative propulsion effects. A hull form study and its optimization were taken as one of possible solutions. Traditional and standard series analysis methods of resistance evolution were not appropriate. The study was carried out using the method of successive approximations and computational fluid dynamics (CFD) algorithms. The flow lines around the hull of the vessel were analyzed and the optimal solution was chosen. Water resistance analysis was carried out to prove the efficiency of the chosen optimization decisions.

1. Introduction

Over the time rising costs of fuel and more stringent environmental requirements [1] make ship owners and designers think of new solutions and structures that will help to reduce operating costs. This leads to a creation of sophisticated design of a ship hull.

Many studies are made in the field of hull form optimization and resistance reduction [2–11]. However, the results of these studies are not always used for creation of hull forms for new vessels. In some other cases, it is not always possible to apply the known methods and their results in solving hull design tasks for new ships. In search of a compromise in ship design some technical and exploitative characteristics deteriorate in order to implement other, not less important, characteristics. Therefore, previous experience and expertise may not always be precisely adapted in new vessel designs. This pushes shipowners to carry out new research with the aim to find the decision where the hull shape and the elements are optimized. One of the optimization objects is the stern. The water flow from the forepart of the vessel should be smoothly flowing around the stern and separating from it. This is particularly important because a poor design of the stern can cause turbulence and wake, which increases the resistance of the hull. All adverse events usually cannot be avoided, so depending on the type of a ship and the projected needs, the stern is designed in an optimal way for working under certain conditions. The stern is also important in the way, that it is usually equipped with propulsion; therefore, its improper design can lead to a negative interaction between the hull and propeller, vibration, cavitation and other unfavourable effects [12–16]. Water flow distribution is also important, because uneven flow to the propeller work field can reduce the effectiveness, thus reducing the speed of the vessel. Optimization of the hull form can reduce hull resistance (wave resistance, improve the flow around the hull) and, in certain cases, can increase the propulsion thrust. Undoubtedly, hull optimization, the research and tests require additional cost and time, but a well-designed vessel will demonstrate higher speed and / or lower fuel consumption, and will result in financial benefits during exploitation. A reliable, well-designed and optimized vessel will be also environmentally friendly due to the fuel economy and reduction in exhaust emissions.

This paper studies the stern of Klaipeda University research vessel. The research vessel is an integral part of the Marine valley – a new complex of scientific and study laboratories aimed at making Klaipeda University a center for marine science and technology in Lithuania [17]. The vessel is used both for research purposes and for various marine sector needs, such as: marine environmental monitoring, fish resource research, respond to pollution incidents and other marine safety related tasks [18]. This ship will contribute to the maritime economy of the planned major infrastructure projects and, on the governmental level, will help to implement the EU Maritime Policy.
goals [19]. The hull type - catamaran, the length of the vessel is 38.7 m; it is equipped with two azimuth thrusters in aft (Fig. 1).

The research objective is to investigate and determine the flow around the stern, to form new stern (skeg) forms on the base of the received data and find the optimal form of the stern. This study aims not only to optimize the stern form, but also to develop general guidelines for creation of stern body shape for such type of vessels, that can be used to avoid design mistakes in the future.

2. Experiment methodology and approach

Ship flows are described by the Navier–Stokes equations, which for the incompressible flow can be written as follows [2]:

$$\frac{\partial V}{\partial t} = \nu \Delta V - \frac{1}{\rho} \nabla p + F,$$

(1)

where $V$ is the velocity vector field, $t$ - time, $\nu$ - kinematic viscosity factor, $\Delta$ - vector Laplace operator, $\nabla$ - nabla operator, $p$ - density, $\rho$ - pressure and $F$ is vector field of mass forces.

One of the weaknesses of CFD is that the results and their accuracy depend of the CFD computer program, the method of calculation, conditions and other parameters. In order to obtain accurate test data it is necessary to create the right conditions for the experimental model.

The experiment was carried out using the CFD computer software FLOW-3D. In consideration of the RANS free-surface methods, there are a number of approaches, dealing with the flow conditions at the location of the air–water interface [2]. The approach, used to determine the location of the water free surface, is capturing the location implicitly through determining where, within the computational domain, the boundary between air and water is located.

Since FLOW-3D approximation accuracy depends mainly on the number of the mesh elements, to create a precise hull form in CFD software will require an input of a particularly dense mesh. In this case, it will result in a significant increase of calculation time and necessary computer resources. Increasing the quantity of mesh cells until certain threshold can make the experiment calculation become too long or even not possible; therefore, an extremely high number of mesh cells is impractical and requires a lot of calculation time. While approximating the hull form, a problem to replicate the sharp edges of the hull arises.

There is a skeg in the aft of the hull of the vessel that has angular edges, therefore approximating it in CFD software will result in some softer edges comparing to reality. Fig. 2 shows the shell approximation with different mesh detailing. By increasing the amount of cell quantity per cubic meter to 4096 (cell size equal 6.25 cm) the computer failed to approximate the hull due to complexity of calculations. The hull model will be also used as the basis for creation of alternative forms for flow optimization around the ship hull.

2.1. Estimation of experiment conditions

At the first stage the optimal boundary sizes for the experiment should be estimated aiming to receive sufficiently accurate results with the feasible calculation time. The experiment is set by the following conditions:

- Maximum vessel speed is 12.5 knots (6.43 m/s);
- Half of the vessel (one hull) is used in the experiment setting symmetry boundary condition in vessel center line;
- Vessel is fixed with the water flowing around it;
- The experiments show, that the ratio between the vessel underwater section area and the experiment basin section area should not exceed 1%, and is recommended to be 0.4% [20].
- The above recommendation, the transversal section of boundary should be around 52 m x 33 m. In this case we have:
  - Large experiment boundaries, higher number of cells;
  - Relatively low detailing of hull form or much longer calculation time with higher cell resolution;

In order to eliminate these disadvantages it is necessary to use a variable mesh, concentrating more cells where the flow details are most important, e.g. around the vessel hull. The experiment is focused on the aftpart of the vessel MINTIS, where the flow around the hull form (skeg area) is the most interesting, providing initial conditions for the efficient work of the propulsion system.

After some preliminary tests, the experiment specifications were developed:

- Symmetry boundary condition will be used;
- Mesh block dimensions in length and height will be not smaller than 33 m. The width of the mesh block must to be also sufficiently big;
- A variable mesh will be used, concentrating the cell quantity near the vessel hull;
- The duration of the experiment must be at least 60 s.
- There are two possibilities for creation of a variable mesh (Fig. 3):
  1. To make the mesh from one block, dividing it into planes, where the size of cells is changing gradually.
  2. To make the mesh from blocks with different cell sizes.

When mesh cell sizes are the same, their volumes are also the same, therefore, the energy transition is stable. When variable-sized cells are used in the mesh, then more complicated equations should be used in order to get sufficiently accurate calculation results. While creating a mesh from several blocks, it is necessary to maintain the ratio between cell sizes. The density of the mesh cells is chosen in such a way, that the edges of different meshes would be aligned. This way CFD software can calculate interaction between different elements faster and more accurately. The scheme of the created mesh is shown in Fig. 4. The number of mesh elements is 4 305 070. The analysis of the experiment conditions showed the main hull hydrodynamic disadvantages, which can be divided into several main groups:

- Decreased flow rate (vortex) at the end of skeg;
- A sudden change of the flow across the lower skeg plane.
- A strong wake, the flow separates disorderly.
- A bow wave partially coincides with a stern wave.

The analysis of flow lines in the area of skeg (Fig. 5) showed that the flow tends to follow the buttock direction, so the direction of flow changes, e.g. switches from the horizontal (bottom of skeg) to the vertical movement following the buttock direction. In this area the flow suddenly changing its direction will become turbulent.
دریافت فوری متن کامل مقاله

امکان دانلود نسخه تمام متن مقالات انگلیسی
امکان دانلود نسخه ترجمه شده مقالات
پذیرش سفارش ترجمه تخصصی
امکان جستجو در آرشیو جامعی از صدها موضوع و هزاران مقاله
امکان دانلود رایگان ۲ صفحه اول هر مقاله
امکان پرداخت اینترنتی با کلیه کارت های عضو شتاب
دانلود فوری مقاله پس از پرداخت آنلاین
پشتیبانی کامل خرید با بهره مندی از سیستم هوشمند رهگیری سفارشات