EXPERIMENTAL AND NUMERICAL BUOYANCY ANALYSIS OF TRACKED MILITARY VEHICLE

Michał Kotowski, Wiesław Barnat

Military University of Technology
Department of Mechanics and Applied Computer Science
Kaliskiego Street 2, 00-908 Warsaw, Poland
tel.: +48 22 683-72-01, fax: +48 22 683-93-55
e-mail: mkotowski@wat.edu.pl, wbarnat@wat.edu.pl

Mirosław Grygorowicz

Gdansk University of Technology
Department of Theory and Ship Design
Narutowicza Street 11/12, 80-233 Gdansk, Poland
tel.: +48 58 347-16-68, fax: +48 58-341-47-12
e-mail: snop@pg.gda.pl

Robert Panowicz, Paweł Dybcio

Military University of Technology
Department of Mechanics and Applied Computer Science
Kaliskiego Street 2, 00-908 Warsaw, Poland
tel.: +48 22 683-73-48, +48 22 683-72-01, fax: +48 22 683-93-55
e-mail: rpanowicz@wat.edu.pl, pdybcio@wat.edu.pl

Abstract

The paper presents experimental validation of numerical flow analysis around tracked amphibious military vehicle. The aim of the study was to validate the simulation results and visualize the flow around optimized geometry To perform numerical simulations, a Computational Fluid Dynamics code FLUENT was used. The software is dedicated to model flow, turbulence, heat transfer, and reactions. The area was divided into two domains – air and water. During the simulation, water was moving with speed specified in standardization. However the numerical techniques are constantly developed and improved there is still a strong need for verification especially in buoyancy analysis and water performance evaluation. Results showed quite good agreement with literature after some modifications of the model. Then the flow was numerically visualized. That allowed in-depth analysis of the flow around each of vehicle's parts and some major and minor modifications were proposed. Modified geometry was analyzed and obtained results will be compared with previous ones. Additional stability and manoeuvrability tests were conducted. The results will be used to modify the geometry of vehicle and conduct experimental tests to verify the numerical code performance and check the actual change in vehicle's water performance. Future work will also consider propeller influence on results.

Keywords: *buoyancy, military vehicle, amphibious vehicle, CFD*

1. Introduction

The aim of presented work was to build and verify numerical model of wheeled armoured vehicle to estimate drag force during movement in water [1]. Such model allows cheaper and faster verification of assumed properties than experimental trials. In-depth analysis of drag versus velocity correlation will allow proper hull and propeller design.

Currently, there are no clearly defined analytical methods determining the drag of vehicle moving in water. The most accurate analysis of the problem conducted shipbuilding industry [2-5], which is based on many years of experience and number of tests carried out in both the laboratory and at the full-size objects. During that time they created a theory which allows moving the results of experiments carried out on the models on the performance of the full-sized units. Unlike military vehicles, ships are characterized by a relatively simple geometry of the submerged part. In addition, they are designed to move only in water environment, which reasons the use of scaling and gives good correlation with the experimental results. Due to the lack of precise analytical methods, it is hard to determine drag force characteristics for military vehicles. It is necessary to perform laboratory tests and verify the results on the real model, which extends the design process and may lead to an overestimation of the properties. The results obtained by the laboratory experiments are subject to certain requirements and approximations [6]. To be able to scale the laboratory results on the geometry of the real it is necessary to use the same movement model description of the vehicle. Usually the Navier-Stockes equation for incompressible fluid is used together with proper boundary conditions. This is due to three conditions. First is the geometric similarity of the vehicle. Second condition is to maintain equal relation speeds of flowing water and the actual vehicle in each pair of corresponding points of space around the vehicle and the actual model. Third condition is equality of dimensionless coefficients of the equation Navier-Stockes for both the model and the actual vehicle. The last of these conditions cannot be met when a laboratory scale model will be other than 1:1 with respect to the actual vehicle. Failure to comply with this condition implies acceptance of certain approximations and scaling factors, generally called scale effects, which affect the accuracy of the results. The use of numerical methods for characterization of drag w allows testing the model of the actual size, without scaling.

2. Numerical model

For the purpose of the study a commercial code Fluent was used. It is dedicated to flow analysis. The environment in which the vehicle is moving numerically is described as Euler area. This area, consisting of 1 600 000 elements, is divided into two mediums, air and water. Tested Vvehicle model is shown in Fig. 1. The geometry of the model is the same as actual length, width and height of the vehicle. Axial symmetry of the model allowed reducing computational domain modelled by one half, which has significantly reduced computation time. The results will be presented for the entire body of the vehicle. It is assumed that the centre of gravity is in the centre of the vehicle.

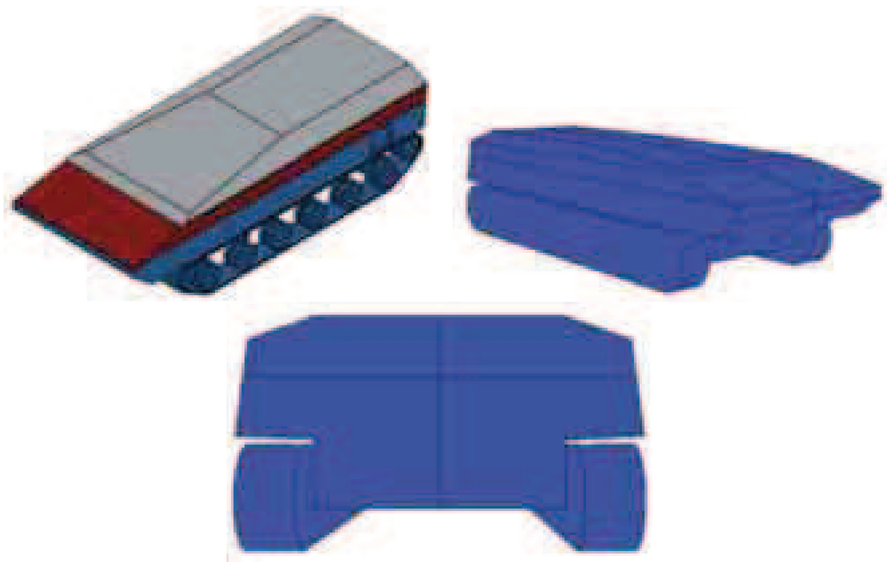

Fig. 1. Numerical model of the vehicle

3. Analysis results

As a result of analysis, fluid velocities and dynamic pressure maps were obtained. Fig. 2. shows velocity in the direction of movement in four layers down from the nominal level of the tracks base. Fig. 3. Shows velocity in perpendicular direction down from the nominal level of the tracks base.

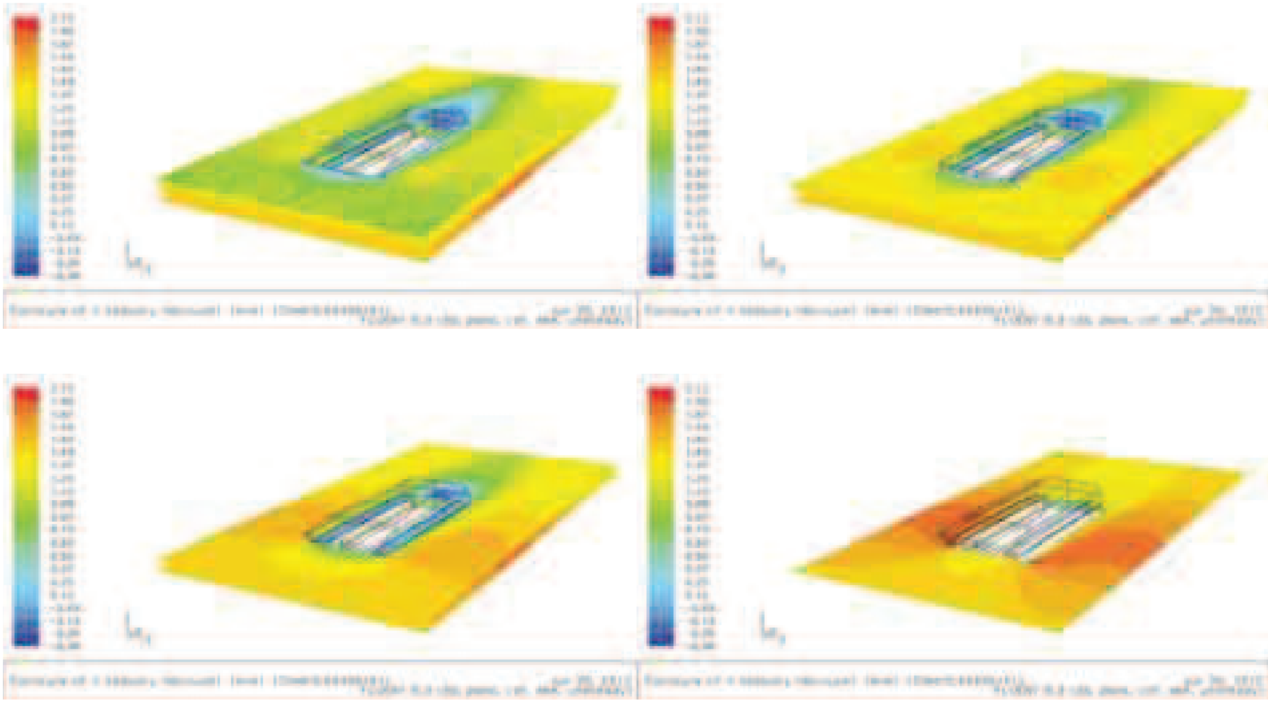

Fig. 2. Velocity of flow 1.5 m/s, velocity in the direction of movement, four layers down from the nominal level of the tracks base

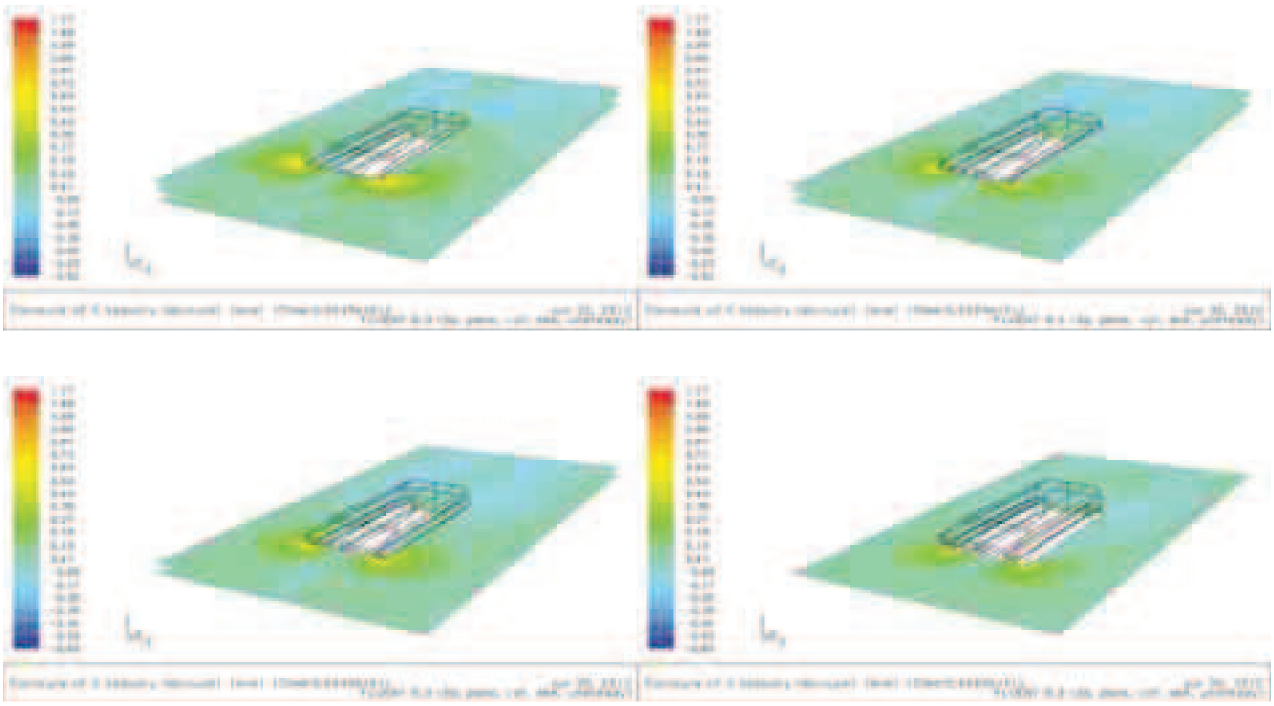

Fig. 3. Velocity of flow 1.5 m/s, velocity perpendicular to the direction of movement, four layers down the from nominal level of the tracks base

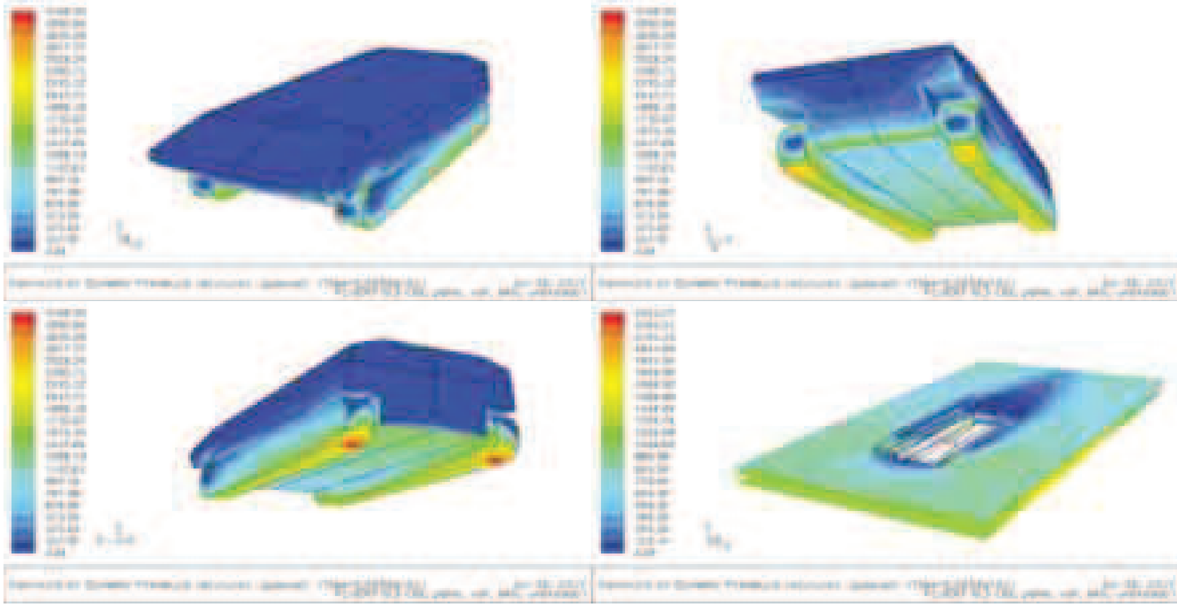

Fig. 4. Velocity of flow 1.5 m/s. Counters of dynamic pressure. Red – accelerated flow, blue – decelerated flow

For a flow velocity of 1.5 m/s more in-depth analysis was carried. It was found that flow has cyclical nature. Maximum, minimum and average drag force which acts on the transporter were read.

4. Conclusions

The paper presents preliminary buoyancy and flow analysis. Presented results will be expanded on a range of flow velocities with different model configurations. As well as that, they will be used to validate the model after experimental trials which are planned after choosing the best hull configuration.

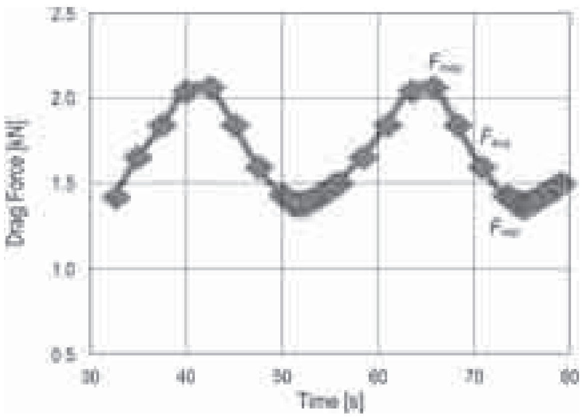

Fig. 5. Drag force vs. time plot

References

[1] Guenter, H. Hohl, *Military terrain vehicles*, Journal of Terramechanics, Vol. 44, Is. 1, 2007.
[2] Poradnik okretowca, Wydawnictwo Morskie, Gdynia 1960.
[3] Wieckiewicz, W., *Budowa kadłubów statków morskich*, Wyższa Szkoła Morska w Gdyni, Gdynia 1999.
[4] Wewiórski, S., *Konstrukcja stalowego kadłuba okrętowego*, Wyd. Morskie, Gdansk 1977.
[5] Bogucki, D., Czarnecki, S., *Geometria kształtu kadłuba*, Biblioteka okrętownictwa, Wydawnictwo Morskie, Gdansk 1983.
[6] Helvacioglu, S., et al., *Improving the river crossing capability of an amphibious vehicle,* Ocean Engineering, Vol. 38, Is. 17–18, 2011.