Mechanical Testing and Diagnosis
ISSN 2247 – 9635, 2017 (VII), Volume 4, pp. 5-8

THE ANALYSIS OF IMMERSION MOVEMENT OF REMOTELY OPERATED VEHICLE

Andra Teodora NEDELCU [1)*], Catalin FAITAR [2)], Nicolae BUZBUCHI [2)]

[1)] Mircea cel Batran Naval Academy, Faculty of Navigation and Naval Transport, 1 Fulgerului, RO-900218, Constanta, Romania
[2)] Maritime University of Constanta, Faculty of Naval Electromechanics, 104 Mircea cel Batran, RO-900663, Constanta, Romania

[*]Corresponding author: andra.nedelcu@anmb.ro

ABSTRACT

Approximately 2/3 Earth surface is covered with water in form of lakes, rivers, seas or oceans. In the recent years, more and more countries have focused and develop the possibility to operate technology and resource exploitation in deep and shallow water. This underwater vehicle helps to improve the ability to inspect and image the seafloor, ensuring a higher resolution of seafloor mapping data. The underwater vehicles may be divided into two types: remotely operated vehicle (ROVs) and autonomous underwater vehicles (AUVs). As opposed to AUVs, which are like the conventional submarines in many aspects, ROVs have different geometries. This is the reason why it is more difficult to analyze the hydrodynamics of ROVs. This paper presents a CFD analysis for an underwater vehicle in submergence, using AnsysCFX and Solidworks to make the vehicle`s graphic design.

Keywords: Underwater vehicle, CFD Analysis, ROV, AUV

1. INTRODUCTION

Approximately 2/3 Earth surface is covered with water in form of lakes, rivers, seas or oceans. In the recent years, more and more countries have focused and develop the possibility to operate technology and resource exploitation in deep and shallow water. This underwater vehicle helps to improve the ability to inspect and image the seafloor ensuring a higher resolution of seafloor mapping data [1].

The underwater vehicle describes both remotely operated and autonomous underwater vehicles. The difference between an autonomous underwater vehicle (AUV) and a remotely operated vehicle (ROV) is that the ROV is connected to a command platform (e.g. a ship) using a tethered cable or an acoustic link. The tethered cable ensures the energy supply and information signals helping the operator onboard of a ship to control and monitor the vehicle all the time. The AUV is equipped with a battery pack and a sonar in order to accomplish its mission without the use of an operator [2].

CFD method is a computational hydrodynamics way to solve Navier-Stokes equation by means of computer-based information using Ansys software [5]. Computers are used to prepare the data, build computational domain and mesh, perform numerical solution of the equations and to analyze the solution results.

The preliminary computer-aided design (CAD) model of an ROV will be created using SolidWorks software (Fig. 1). In order to obtain CAD model and hydrodynamic parameters in immersion like hydrodynamic damping, pressure and forces, the authors use Ansys CFX software.

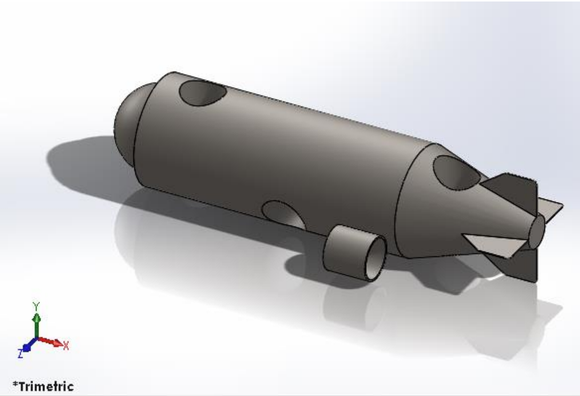

Fig. 1. Model design in SolidWorks software

Figure 2 presents the chart of proposal systematic computation of ROV model from numerical modeling to control system design.

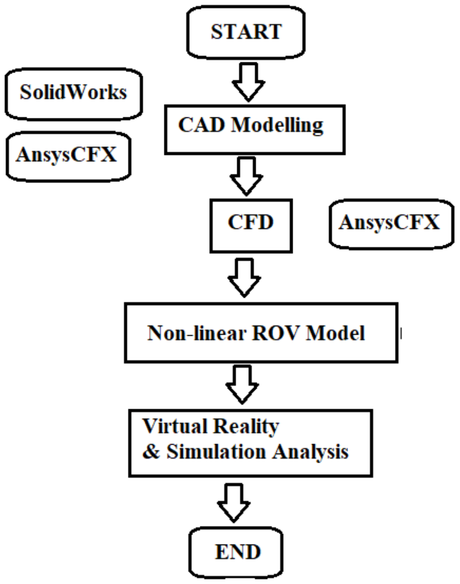

Fig. 2. Chart of proposal systematic computation of ROV model for virtual reality

2. SIMULATION

Using ANSYS CFX software, in this case, the number of elements is approximately 90769. The fluid domain has around 497601 nodes. The volume is 2232.9 m^3. The 3D view of volumetric mesh of the flow domain around ROV is represented in Fig. 3.

Figure 4 presents the boundary conditions. The authors chose a “no-slip” condition because water can not penetrate through the solid surface. The flow regime inside the sphere is subsonic, the vehicle reaching a velocity of 2 m/s during the simulation. Other initial conditions are: water temperature 25°C and a turbulent flow.

Figure 5 presents the velocity of vehicle when is in immersion. The authors observe a high value of velocity on the superior part of vehicle and a small one on the inferior part. The streamline has an irregular trajectory and shape, started with bow, and ended on the stern.

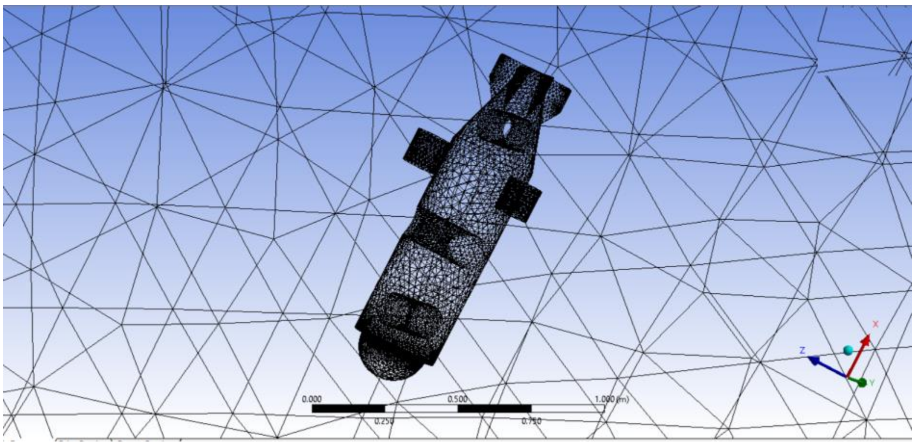

Fig. 3. 3D model design in SolidWorkds and imported in Ansys CFD

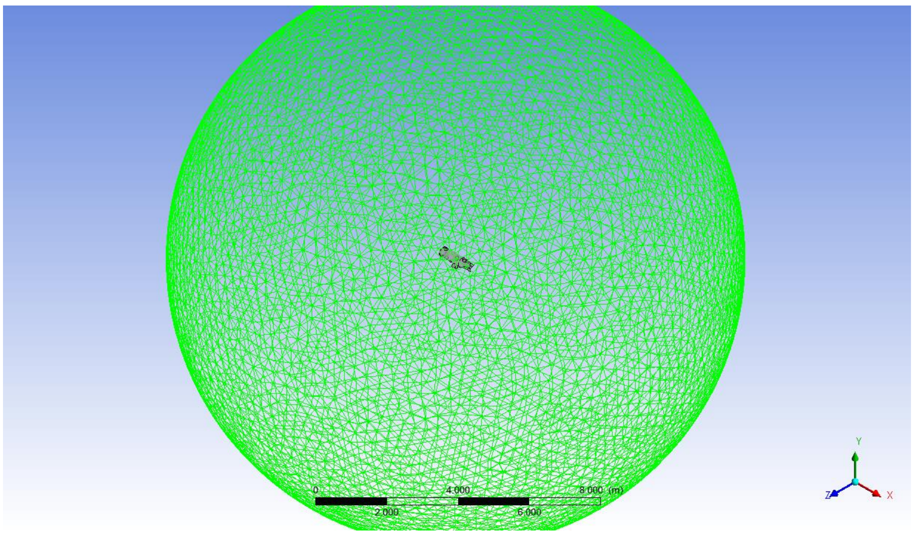

Fig. 4. The vehicle inside the domain

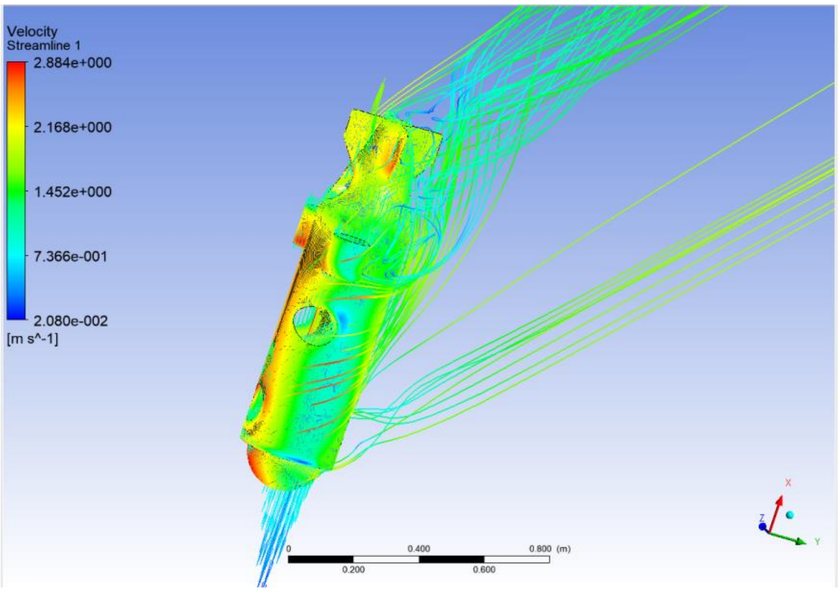

Fig. 5. The velocity streamline when vehicle is in immersion

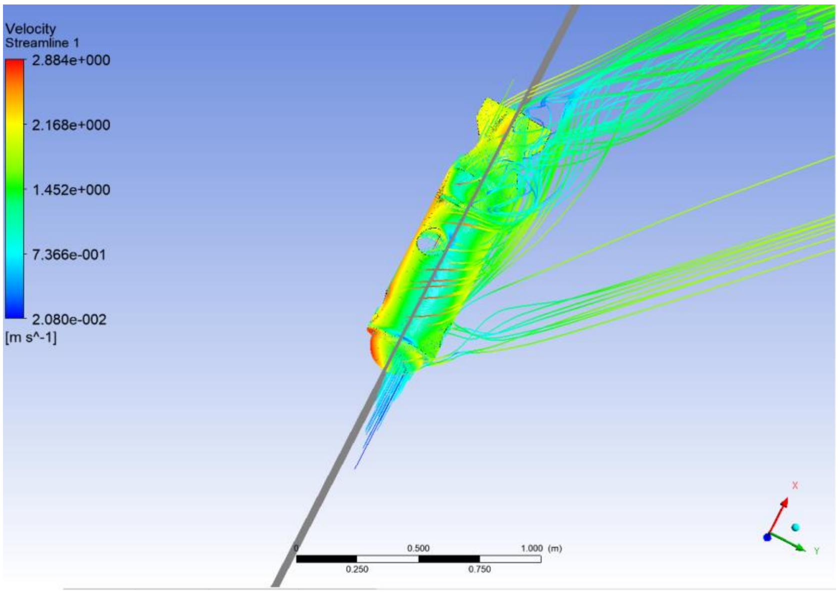

Fig. 6. Longitudinal section over vehicle body

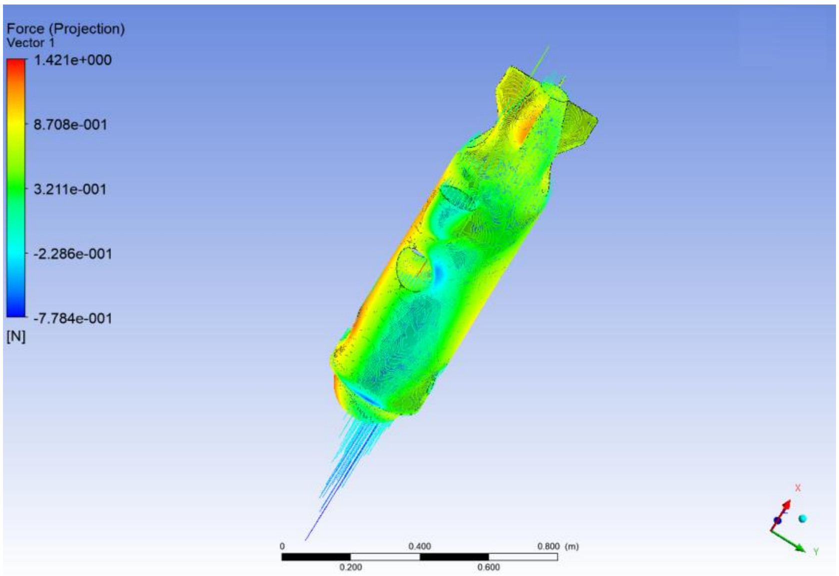

Fig. 7. The vector force over vehicle

Figure 6 presents the vehicle in immersion. Along the vehicle body, a longitudinal plan was introduced in order to see much better the difference between the top part and the underside part of the vehicle. The number of streamline is bigger on underside part in contrast with top side. Also, the vehicle achieves a maximum velocity value 2,884 m/s on top side due to water advancing.

The vehicle achieves a small force value of -7.784 kN on bow, and a value of 8.7 kN on stern (see Figure 7). The maximum force value is on top of vehicle, near the holes for propulsion.

3. CONCLUSION

The simulated result is going to be verified in the following with the experimental tests in basic and sea water. Future works could improve the accuracy of the CFD results by comparing the numerical simulation with the ROV using real-time adaptive identification approach in sea trial

REFERENCES

[1] Committee on Autonomous Vehicles in Support of Naval Operations Naval Studies Board Division on Engineering and Physical Sciences, Autonomous Vehicles in Support of Naval Operations, 2005.

[2] Christ R. D., Wernli R. L., Sr., The ROV Manual, A User Guide for Remotely Operated Vehicles, Second Edition, Butterworth-Heinemann 2014.

[3] Rusu E., Mecanica, Editura Universității Dunărea de Jos, 2001.

[4] Rusu E., Numerical Simulations of the Current Field In The Black Sea Basin, Constanta Maritime University Annals 19, 2013, https://trid.trb.org/view/1309931

[5] Ansys CFX software, https://caeai.com/ansys-software-support/ansys-software/computational-fluid-dynamics-ansys-cfx-and-fluent-cfd-software