The vehicle movement using computational fluid dynamics method

A-T Nedelcu¹, C Faita², L Stan², N Buzbuci² and L M Dumitrache²
¹Mircea cel Batran Naval Academy, Faculty of Navigation and Naval Transport, 1 Fulgerului Street, 900218 Constanta, Romania
²Maritime University of Constanta, Faculty of Naval Electromechanics, 104 Mircea cel Batran Street, 900663, Constanta, Romania

E-mail: andra.nedelcu@anmb.ro

Abstract: Underwater vehicles represent underwater robots able to perform tasks without or with a minimum human intervention. Research and development of this vehicles have growing as a result of excellent characteristics operated in different situation. Before discussing the steady motion of underwater vehicle, it is necessary to examine the hydrometeorological condition founded in respectively area. Interaction between the water surface and the atmosphere determine the development and appearance of hydrodynamic processes. The main current encountered in Black Sea Basin moves in a cyclonic direction to the edge of the continental platform and surrounds the entire basin. Using the data in-situ regarding the influence of marine current for different steps of depths, the underwater vehicle reacts different and have to change his input data to follow the right course. This paper provides an intuitive modeling and simulation approach to obtain a distribution of current influence along the underwater vehicle body. The software SolidWorks and Ansys Fluent are used to realize the vehicle design and the simulation. The Ansys software is a powerful tool used for solving problems involving fluid mechanis. The condition imposed for simulation, was generated and determinate in-situ by Current Meter Model 106.

1. Introduction
The underwater vehicle has a large range of applications in marine geosciences. Their interest is increasingly also in scientific, military, commercial and policy sectors. They are well suited to exploration of extreme environments, from the world’s deepest hydrothermal vents to beneath polar ice sheets.

Over the last few decades, ocean research and exploration have made underwater mechanical systems a necessity. Underwater vehicles provide a new kind of marine platforms that could represent a great necessity in many areas of oceanographic research. Until now, the underwater vehicles come in a verity of shapes, sizes and means of propulsion. Depending on these characteristics, the type and mission of vehicle are also determined.

The primary applications in marine geosciences that have resulted from underwater vehicle are the studies of submarine volcanism and hydrothermal vent, mapping and monitoring of low-temperature fluid escape features and chemosynthetic ecosystems and benthic habitat mapping in shallow and deep-water environments.

To specify realistic operational requirements of an underwater vehicle, we must take into consideration some oceanography and meteorology condition. In the design of underwater vehicle, an
important role is represented by the properties of the ocean floor, water itself or the atmosphere above [1]. Usually, one is concerned with the interfaces between these zones- firstly that zone at the sea surface and the transition from one zone to another takes place abruptly. For example, the mean density of the sea bottom is around 2.5 Mg m$^{-3}$, that of the sea water is around 1.03 Mg m$^{-3}$ and the mean density of air at sea level is 1.23 Mg m$^{-3}$.

2. Circulation of currents in the Black Sea

Interaction between the surface of water and the atmosphere leads to the creation of hydrodynamic processes in the marine environment.

These energy exchanges and hydrodynamic processes aim is to create sea currents and waves. In both cases, the motion is determined by the tangential interaction of wind from the sea surface that turns into quasi-motion in active layers and wave ondulatory on the surface of water. Marine currents have a noticeable influence on the physico-chemical and hydrodynamic conditions of marine and ocean waters. The formation of currents is based on two causes. The main cause behind the formation of currents is winds that give rise to "drift currents". Another determining cause for the formation of currents is due to the differences in density on different depths, which leads to formation of superficial currents, called "convection currents".

The main current encountered in the Black Sea Basin (Rim current) has a move in the cyclonic direction to the edge of the continental platform and surrounds the entire basin.

Over several time have been determinated different schemes of surface current in the Black Sea studied by different authors: the "dynamic method" [2, 3] achieved by applying measurements using instruments; and quasi-geostrophic model with data up to 1982 [4]. Similar patterns of circulation are also presented in monographs [5].

3. In-situ measurements

The measurements made during the year 2018 were executed on two decks along the Romanian seaside. One of them was made during the summer at the coordinate point 44°13'54.5" N 28°38'03.1" E. The second measurement was carried out in winter at 44°09'21.0" N 28°39'54.0" E. For both measurements, was used Model 106 Currentmeter (figure 1).

![Currentmeter Model 106](image)

**Figure 1.** Currentmeter Model 106.

The instrument used for both measurements is designed to determine real-time water parameter values. It works at every 1 second cycle, at which time only one compass path value is recorded. To do this, the eastern and northern speeds are calculated, and then summed for an average period. Additional parameters that it offers are temperature and pressure. Each measurement started at a
depth of 2 meters and ended at 6 meters, except for the first measurement where the depth of sea was 3.25 meters.

The simulations were performed for the vehicle body position at different depths within the range so that the vehicle simulates different pressure ranges starting from 0.2 bar (corresponding to a depth of 1 meter) to 0.6 bar (corresponding to a depth of 6 meters). The results obtained are presented in figure 2.

Depending on the depth at which the simulation is executed, the Currentmeter Model 106 has different values of direction and velocity of current. Measurements made with current meter took place in two stages: the first stage during the summer, when according to Graphic, the current velocity is relatively small.

The second stage of simulations was carried out during the winter, when the current speed reaches maximum values. The device recorded data from a depth of 2 meters to about 6.5 meters.

The simulations performed with the Ansys software have been done in various areas with varying depths, with the underwater body simulating speeds between 2 and 10 knots.

4. Determination of vehicle motion equations

It is known the movement of underwater vehicle if at any time it is possible to determine and find its position in space, [6]. At the same time, the position of vehicle in space is determined by the generalization of six coordinates according to figure 2:

a) Mass Center Coordinates: $X_{ag}$, $Y_{ag}$, $Z_{ag}$

b) Angles:
- Yaw angle between the $OgXg$ axis and the longitudinal axis of vehicle;
- The angle of trim between the horizontal plane $XgOgZg$ and the longitudinal axis of vehicle;
- Roll angle, between the $YgOgXg$ vertical plane and the vehicle's diametrical plane, $XOY$
Figure 3. The coordinate system for study the vehicle motion.

In the speed coordinate system, the position of vehicle is determined by angle of attack and angle of drift.

The angle of attack, $\alpha$, is the projection of velocity vector on the diametrical plane and longitudinal axis of vehicle;

The drift angle, $\beta$, is measured between speed vector and vehicle’s diametrical plane.

The linear and angular speed of vehicle is represented by the fixed body and is derived from equation (1):

$$u = [u, v, w]^T \in \mathbb{R}^3, \quad \omega = [p, q, r]^T \in \mathbb{R}^3$$

Where $u$ and $\omega$ represent the linear and angular velocity.

The position and angle of inclination (Euler angle) of vehicle is generally expressed according to a fixed coordinate system. In this paper, a NW coordinate system was chosen in which the x-axis represents the true north direction, the Y-axis direction west, the vehicle moving towards the west direction, and the z-axis representing the normal Earth surface. Euler's position and Euler angle of vehicle are determined by the vector $\eta = [\eta_1^T, \eta_2^T]^T$ where each parameters are determined by equation (2) and equation (3):

$$\eta_1 = [x, y, z]^T \in \mathbb{R}^3$$

$$\eta_2 = [\phi, \theta, \psi]^T \in \mathbb{R}^3$$

The underwater vehicle when immersed in water forms a complex hydrodynamic system together with the liquid around it. Compared to the movement of bodies in the air, for the underwater vehicle motion equations, account shall also be taken of the characteristics of the water surrounding the body of vehicle.

From the point of view of the liquid areas surrounding the body of vehicle, they can be subdivided into [7]:

1) Limit layer: representing the area where forces due to viscosity are manifested. Here the liquid is considered real and compressible.
2) The slipstream, or the turbulent flow: representing the volume of fluid which leaves behind a body moving in a fluid, whereas a certain amount of fluid particle velocity from the vehicle, its displacement cannot be achieved rather than changing the trajectory. In case of hydrodynamic shape bodies, the point of separation of the boundary layer (from where the turbulent flow starts) is in the region of the tail float (truncated segment).

3) Outside potential flow area: Liquid is considered ideal in this area. Because the thickness of the boundary layer is relatively small and the liquid in the third zone influences the movement of vehicle in water, the layer becomes thinner when the speed increases.

In the third area the liquid is studied according to the method of potential movements, according to equation (4):

\[ \mathbf{v} = \nabla \phi \]  

(4)

Where \( \nabla \phi = \frac{\partial \phi}{\partial x} \mathbf{i} + \frac{\partial \phi}{\partial y} \mathbf{j} + \frac{\partial \phi}{\partial z} \mathbf{k} \), and where \( \phi \) represent potential function.

Beyond the boundary layer boundary, in the area of the potential outside flow, the liquid can be considered as ideal, viscous. We will then analyze the influence of the ideal flow on the energy state of the vehicle and, implicitly, on the character of its movement.

If the vehicle moves at a constant speed, then the amount of motion (impulse) induced to the particles of the fluid environment will have a constant value. Taking into account the momentum variance theory and the kinetic momentum variation theorem, the energy action of the water on the vehicle will be null. In the case of its unstable movement, the variation in motion quantity is of a permanent character, determining the occurrence of forces and moments of inertial nature. Another explanation can be given because the vehicle’s pressure on the ideal liquid must only cancel out its inertia, with frictional forces being absent in this area of analysis.

The underlying hypothesis that underlies the study of the phenomenon presented claims that the underwater vehicle in motion, together with the volume of fluid acting on it, forms a complex hydrodynamic system. In other words, the movement of the center of gravity of the vehicle on the desired trajectory must take into account the hydrodynamic "behavior" of both the vehicle and its external environment. Under these conditions, the theoretical momentum and kinetic momentum will have the equation (5):

\[ \frac{d(\mathbf{H}+\mathbf{Hl})}{dt} = \mathbf{F} = \frac{d(\mathbf{K}+\mathbf{Kl})}{dt} = \mathbf{M} \]  

(5)

\( \mathbf{H}, \mathbf{Hl} \) represents the vector pulses of the vehicle body, respectively, of the liquid expressed in the fixed reference system;

\( \mathbf{K}, \mathbf{Kl} \) are the vectors of the kinetic momentum of the body of the vehicle and of that liquid, determined in the fixed reference system;

\( \mathbf{F} \) is the result of the external forces acting on the hydrodynamic system;

\( \mathbf{M} \) is the result of moments of external forces

Equation (4) can be used to analyze motion of the vehicle in the ideal fluid, provided that the forces and extra moments acting in this case are included in the vectors \( \mathbf{F} \) and \( \mathbf{M} \).

Another way to write equation (4) is in the connected system (5). In this case, there is a simplification of the form of the equation (6):

\[ \mathbf{a} = \mathbf{a}_x \mathbf{i} + \mathbf{a}_y \mathbf{j} + \mathbf{a}_z \mathbf{k} \]  

(6)

The associated coordinate system moves in \( dt \) with the \( v dt \) value. It follows that the additional change of the moment will be equal to \( \langle v dt \rangle \mathbf{F} \), and the speed of modifying the moment \( \mathbf{vxF} \) based on the expression (1) can be written as equation (7) and equation (8):

\[ \frac{d(\mathbf{H}+\mathbf{Hl})}{dt} = \frac{d(\mathbf{K}+\mathbf{Kl})}{dt} + \omega \times (\mathbf{H} + \mathbf{Hl}) = \mathbf{F} \]  

(7)
\[ \frac{d(K + KL)}{dt} = \frac{\partial (K + KL)}{\partial t} + \omega \times (K + KL) + \nu x (H + HL) = M \]  

(8)

Designed on the related reference system will determine the equation (9):

\[ \frac{d(H_x + H_x)}{dt} + \omega_y (H_x + H_x) - \omega_x (H_y + H_y) = F_x \]

\[ \frac{d(H_y + H_y)}{dt} + \omega_z (H_x + H_x) - \omega_x (H_y + H_y) = F_y \]

\[ \frac{d(H_z + H_z)}{dt} + \omega_x (H_y + H_y) - \omega_z (H_z + H_z) = F_z \]  

(9)

To determine the moments functions in the directions OX, OY, OZ we have the equation (10):

\[ \frac{d(K_x + K_x)}{dt} + \omega_y (K_x + K_x) - \omega_x (K_y + K_y) + \nu_y (H_x + H_x) - \nu_x (H_y + H_y) = M_x \]

\[ \frac{d(K_y + K_y)}{dt} + \omega_z (K_x + K_x) - \omega_x (K_y + K_y) + \nu_z (H_x + H_x) - \nu_y (H_y + H_y) = M_y \]

\[ \frac{d(K_z + K_z)}{dt} + \omega_x (K_y + K_y) - \omega_z (K_z + K_z) + \nu_x (H_y + H_y) - \nu_y (H_z + H_z) = M_z \]  

(10)

5. Underwater vehicle graphic design

5.1. Vehicle geometry and domain limits

Figure 4 schematically illustrates the geometry of the vehicle, along with the limits of the computing domain. The body of the vehicle can be divided into three sections: the bow section in the form of a semi-sphere, the actual body of the vehicle in the form of a cylinder and the aft body represented by a conical cylinder [8]. The 3D cartesian coordinate system used in the study is implemented by initially placing the origin of the coordinates in the vehicle’s forward position, then being placed along the body of the vehicle to draw the propellers, and finally into the suture for the wings drawing. The OX axis is taken along the length of the vehicle. Therefore, in the first stage, all axial distances along the body of the vehicle are calculated in relation to the front of the vehicle.

![Figure 4. Vehicle geometry and domain limits.](image)

The main dimensions of the underwater vehicle are shown in table 1.

Table 1. The main dimensions of the underwater vehicle.

| Physical features | Values | Units |
|-------------------|--------|-------|
| Length            | 1200   | [mm]  |
| Diameter          | 300    | [mm]  |
Propeller diameter 120 [mm]
Number of propellers 5 [piece]
The length of the fins 165 [mm]
Number of fins 4 [piece]
Diameter of the tail 90 [mm]

5.2. Vehicle geometry construction steps
To build the underwater vehicle design was used SolidWorks software.

Vehicle body geometry construction steps (figure 5) using the SolidWorks software were as follows:
1) Create a Sketch with the body dimensions of the vehicle;
2) Doubling this Sketch in the OY direction to generate the lower body along the OX axis;
3) Creating mirrors on both sides of the vehicle body using Mirror function;
4) Create a new plan to achieve the tail of the vehicle;
5) The CirPattern function was used to make the fins from the tail of the vehicle;
6) The Revolve function was used to fill the vehicle on either side of the axle;
7) Creating holes on the body of the vehicle using the Cut-Extrude function;

Figure 5. Vehicle body geometry construction steps using the SolidWorks.

5.3. Simulation condition
After design the vehicle body using SolidWorks software, simulations were made using the Ansys software.

The environment in which the underwater vehicle is moving is known as the domain. To achieve the simulation, a wide enough range has been created to avoid that the field limits affect the flow along the body of the vehicle. The dimensions regarding the range (length, width, depth) around the underwater vehicle are presented in figure 6.
Numerical determinations were considered to be a parallelepipedal domain that fits the body of the vehicle.

In order to observe the flow phenomenon in simulations, the density of the node network was more concentrated in certain regions of the domain. In this respect, a fine mesh on the entire body surface of the vehicle was used to solve the flow in the boundary layer.

Generation of the fluid domain network was done in three steps:
1) A free mesh on the surface of the fluid range was created to automatically determine a suitable number of divisions on each face edge using the Face Meshing function;
2) The node network has been refined using the Tetrahedrons method using the Patch Conforming function;
3) Cell layers were added to the body of the vehicle using the Inflation function to capture flow in the boundary layer.

To measure the quality of the generated network, the Skewness and Orthogonal quality functions in Mesh Statistics - Mesh Metric, these being the most important methods used to determinete network quality.

Thus, a mesh comprising 163285 nodes and 912136 elements was formed.

Simulations were performed to position the vehicle at a single depth and speed, and the modeling included the following assumptions and simplifications:
- Homogeneous flow of incompressible fluid;
- Linear motion with constant speed;
- Free area;
- Right fund without natural disturbances;
- No dynamic mesh;
- The geometric modeling of vehicle has been done on a natural scale;
- Fluid flow was carried out in a parallelepipedal area around the body of vehicle.

6. Modeling the flow around the body of vehicle
The position of vehicle varies in the direction of OY axis, the simulations being made for different positions of vehicle to the seabed.

The movement is considered stationary and subsonic. For modeling, heat exchange and weight forces were not considered. The wall rust was considered to be 5 μm.

Simulations have been made for a total vehicle speed and current acting on its body of 10 m/s at the first step, and then 2m/s, 25 degree ambient temperature. The vehicle is at a depth of 7 meters.

Figure 7 shows the total pressure exerted on the body of vehicle. Coincidence of simulation, the maximum value of pressure is 50891.4 Pa, reached in the front of vehicle, according to the legend
where is the red color. The minimum value is -92984.5 Pa reached in the immediate vicinity of vehicle's coat in the slot of coat, it is materialized in figure 6 with the blue color.

![Figure 7. Total pressure exerted on the body of vehicle.](image)

We consider a line passing through the center of vehicle. There are two broken portions of line. This is due to the presence of holes in the crosswise planes on the body of vehicle.

Figure 8 shows the graph resulting from the pressure simulation along the line created. The pressure on the surface of body is constantly on its face, then an area in which it disappears due to direct contact with the environment. The graph is presented according to the length of vehicle and the pressure distribution determined by the line that crosses the body.

![Figure 8. The graph resulting along the lateral line.](image)
The pressure distribution on the transverse surface of body is shown in figure 9. There is a peak pressure zone in the vehicle, where it reaches 50891.4 Pa following a minimum area in the vicinity of coffin and an average area marked with the green color near the coffin, in front of vertical propulsors and the aft of vehicle.

![Figure 9. The pressure distribution on the transverse surface of body.](image)

The resulting pressure on the transverse surface of vehicle body is shown in figure 10. This pressure is exemplified on the entire body surface of vehicle. Including the graph shows the area of maximum and minimum pressure across the entire cross-sectional area.

![Figure 10. The graphic along the total surface of body vehicle in longitudinal plan.](image)
In the second part of simulation, the vehicle speed is 2 m/s. According to figure 1, we have a maximum value of pressure in front of body vehicle. The pressure is lower than the case of 10 m/s speed. But, we have the same position of maximum and minimum value of pressure along the body vehicle.

![Figure 1](image1.png)

**Figure 1.** Pressure along the vehicle at 2 m/s.

According to figure 12 and figure 10, we have some difference regarding the total pressure along the body vehicle. This pressure is influenced by the speed of vehicle. When vehicle depart with 10 m/s, he reached more pressure in front of body comparative when he depart with 2 m/s.

Along the body, the total pressure is the same in both cases.

![Figure 12](image2.png)

**Figure 12.** The graphic with Total pressure at 2m/s speed.
7. Conclusions
Analyses of the effect of underwater current in two different points on Black Sea basin was performed, using the Currentmeter Model 106. The result obtained in different seasonal condition was discuses in this paper. Also, in this analyses we also considered that the underwater current could influenced the movement of underwater vehicle.

With the simulation structure built, two separate simulations were executed with different velocity. Added to velocity was the current value (the biggest value of current was added to velocity value). The purpose of two simulations was to simulate the total pressure along the body vehicle of different speed in the moment when the vehicle crosses a current area in Black Sea basin. The simulation results showed that in both cases the highest pressure value is located in front of the vehicle. We consider a line passing through the center of vehicle. The pressure on the surface of body is constantly on its face, then an area in which it disappears due to direct contact with the environment. There is a peak pressure zone in the vehicle, where it reaches 50891.4 Pa following a minimum area in the vicinity of coffin and an average area marked with the green color near the coffin, in front of vertical propulsors and the aft of vehicle. The current values in the Black Sea basin do not record very high values (0.02-0.08 m/s). There will be no major changes on the surface of the body of the underwater vehicle to alter the set trajectory.

8. References
[1] Ivanov V A and Belokopytov V N 2013 Oceanography of the Black Sea National Academy of Sciences of Ukraine Marine Hydrophysical Institute Sevastopol p 100-134
[2] Neumann G 1942 Die absolute Topographie des physikalischen Meeresniveaus und die Oberflächenströmungen des Schwarzen Meeres Ann D Hydr Mar Met 70 265
[3] Ozsoy E and Unluata U 1997 Oceanography of the Black Sea: A review of some recent results Earth Sci Rev 42(4) 231–272
[4] Oguz T, Latun V S, Latif M A, Vladimirov V V, Sur H I, Makarov A A, Ozsoy E, Kotovshchikov B B, Eremeev V and Unluata U 1993 Circulation in the surface and intermediate layers of the Black Sea Deep Sea Res 40(Pt1) 1597–1612
[5] Filippov D M 1965 The Cold Intermediate Layer in the Black Sea Oceanology 5(1) 47-52
[6] Nedelcu A- T, T O , Clinci C and Ichimoaiei G 2018 CFD approach used for modelling hydrodynamic analysis and motion characteristics of a remotely operated vehicle IOP Conf. Series: Earth and Environmental Science 172
[7] Tarabuta O 2010 Principiile constructive ale vehiculelor subacvatice teledirijate prin fir (Constanta: Publishing House of Academiei Navale) p 30-50
[8] Christ R, R WERNIL AND Sr 2013 The ROV Manual- A user Guide for Remotely Operated Vehicles. Second Edition p 20-50