CFD approach used for modelling hydrodynamic analysis and motion characteristics of a remotely operated vehicle

A Nedelcu\textsuperscript{1}, O Tărăbuță\textsuperscript{2}, C Clinci\textsuperscript{3}, G Ichimoaiei\textsuperscript{4}

\textsuperscript{1}PhD Student, ” Mircea cel Bătrân” Naval Academy, Constanta, Romania
\textsuperscript{2}Assistant professor PhD., ” Mircea cel Bătrân” Naval Academy, Constanta, Romania
\textsuperscript{3}Lecturer PhD, ” Mircea cel Bătrân” Naval Academy, Constanta, Romania
\textsuperscript{4}Lecturer PhD, ” Mircea cel Bătrân” Naval Academy, Constanta, Romania

andra.nedelcu@anmb.ro
octavian.tarabuta@anmb.ro
catalin.clinci@anmb.ro
gheorghe.ichimoaiei@anmb.ro

Corresponding autor: andra.nedelcu@anmb.ro

Abstract. Towed underwater vehicles represents essential tools for providing safe access to the underwater world with many applications including ocean research, naval operations, inspection and repair different of undersea structures. Recently, there is a necessity to use smaller underwater vehicles, tethered or untethered by ship in rivers, lakes or oceans. This underwater vehicle represents an integral part of scientific equipment to explore the seas and oceans. Remotely operated vehicle (ROV) is one of these vehicles which consists of a surface body, connecting cable and underwater vehicle. The dynamic behavior of Remotely operated vehicle (translational and rotational motion or velocity, also sum of all forces acting on the system) is numerically investigated. This paper provided a model used for simulation and modeling approach to obtain the hydrodynamic characteristics of an underwater vehicle using computational fluid dynamic (performed by the Ansys CFX). This is the preliminary design stage where extensive hydrodynamic test facilities are not available.

1. Introduction

The remotely operated vehicles are used mostly in the exploration of ocean resources. They come to help humans to investigate in a safety mode the deep and hostile underwater environments without their physical presence. They have different applications including naval operations (Banerjee and Do, 1995; Ohkus et al., 1987; Park et al., 2003), ocean research and exploration (Boe et al., 2013; Feng and Allen, 2004), acoustic surveying (Buckham et al., 2003), assembly, inspection and repair of undersea structures (Kamali and Khojasteh, 2016; Khojasteh, 2015). ROV is a towed underwater vehicle. The system is composed of the underwater vehicle, which is connected to the water surface on a vessel, buoy or platform, a handling system and other associated power supplies. The vehicle is linked to the ship using an umbilical cable. The vehicles can very depend on size: from small vehicles with video cameras for simple observation up to complex systems, which can have several and different equipment’s, tools and other manipulators. Take into consideration the importance of
vehicles applications nowadays has been made many applications and experiments to examine the
dynamics of underwater vehicle, towing cable and surface vehicle.

2. **Computational fluid dynamic analysis of an experimental underwater body vehicle**

In this paper, the principal objective is to establish and determinate a CFD methodology for
the computation of the drag and lift forces on body vehicle. The behavior of these forces gives the
hydrodynamic coefficients of the body vehicle that are required for its trajectory simulation. As a
result, an accurate estimation of the hydrodynamic lift and drag forces is critical in studying body
vehicle trajectories regarding a variety of operating conditions. Since no experimental measurements
of these hydrodynamic forces have not been performed in the present work, it is important to validate
the CFD methodology with experimental data and associated phenomena by means of computer-based
simulation.

The hydrodynamic forces on the body vehicle depend on the body construction. We
considered the body vehicle used for CFD analysis for estimating the drag and lift forces for different
speed combinations (in Figure 8). The design particulars are presented in Table 1.

| Physical property   | Value | Units |
|---------------------|-------|-------|
| Length overall      | 1200  | [mm]  |
| Diameter of body    | 300   | [mm]  |
| Diameter of propellers | 120 | [mm]  |
| Number of propellers | 5    | [piece] |
| Length of wings     | 165   | [mm]  |
| Number of wings     | 4     | [piece] |
| Diameter of tail    | 90    | [mm]  |

The preliminary computer-aided design (CAD) model of an ROV will be created using
SolidWorks software. Through CAD model we obtained the hydrodynamic parameters like
hydrodynamic damping, pressure and forces. For this, we use Ansys CFX software.

In order to achieve the ROV modeling approach, Figure 1 presents the overall approach from
numerical modeling to control system design.

![Diagram](image)

**Figure.1. Chart of proposal systematic computation of ROV model for virtual reality**

In the Figure 2 it is presented the geometrical model (front, left, top and right side) of the ROV
vehicle after modeled by SolidWorks. The design particulars are presented in Table 1. The ROV
presents six degrees of freedom for the body position and speed.

The computational domain around the experimental hull body shown in Figure 3 it is
represented by a sphere with a radius of 7.5 m and 1 767,15 m³ volume.
The vehicle is situated along OX axis. We consider the domain (the sphere) full of water with the follow condition presented in Tabel 2:

**Tabel 2. Fluid domain data**

| Fluid domain data | Value | Units     |
|-------------------|-------|-----------|
| Density           | 997.0 | [kg/m³]   |
| Molar Mass        | 18.02 | [kg/kmol] |
### Table 1: Physical Properties of ROV Water

| Property                      | Value      | Units          |
|-------------------------------|------------|----------------|
| Specific Heat Capacity        | 4181.7     | [J/kgK]        |
| Reference Pressure            | 1          | [atm]          |
| Reference Temperature         | 25         | [°C]           |
| Dynamic Viscosity             | 8.899E-4   | [kg/ms]        |
| Thermal Conductivity          | 0.6069     | [W/mK]         |
| Absorption Coefficient        | 1.0        | [1/m]          |
| Thermal Expansivity           | 2.57E-04   | [1/K]          |

To simplify the dynamic equation derivation of ROV, we consider the following properties for the model:
- Operational at slow speed (approximate 2m/s);
- The vehicle body must be rigid and fully submerged in water (without wave and current disturbance);
- The tether (or umbilical cable) dynamic attached to ROV body is not considered.

### 3. Theoretical Issue

#### 3.1. Navier-Stokes model for laminar motion

In the following will be related a few physical, mathematical and numerical aspects about the Navier-Stokes equations system.

**Physical aspects.**

The movement of a fluid is determined by the interaction of many parameters. An essential role is played by the ratio of two types of forces: inertial forces and respectively, friction forces. This report is highlighted by the Reynolds number. Depending on Reynolds number, we can distinguish three flow regimes: laminar regime (for small Reynolds number), turbulent regime (high Reynolds number) and transient regime (between laminar and turbulent flow regimes). In conformity with flow regime, the appearance of movement is very different. For example, for a laminar flow, when the friction forces are dominant, the motion has a perfectly ordered appearance. Contrariwise, for a turbulent flow, when the inertial forces are dominant, the aspect of the movement become completely chaotic, because this chaotic character is mostly highlighted by large variations in time and space of all motion parameters (speed, pressure field, etc.).

The movement can reach different values according to the flow regime. In case of a laminar flow, when the friction forces are dominant, the motion has a perfectly ordered appearance, for a turbulent flow, (dominant inertial forces), the aspect of the movement becomes completely chaotic. The chaotic character is highlighted by large variations in time and space of all motion parameters (speed, pressure field, etc.).

**Mathematical aspects.**

The Navier-Stokes non-stationary equations represent a system of non-linear partial-derivate equations of the second order. Nonlinearity is a result of inertial forces. If inertial (destabilizing) forces are small according to friction (viscous diffusion plays a stabilizing role), the nonlinearity is relatively weak, and the respective equations can be numerically integrated without further hypothesis. According to the above, this case corresponds to the laminar flow regime.

**Numeric aspects.**

The basic problem in the numerical calculation of turbulent flows is the coexistence of structures characterized by time scales, velocities and lengths of different orders of magnitude: from scales corresponding to the general convection movement, to scales with several orders of smaller size characteristic of turbulent or viscous diffusion.

**Navier-Stokes equation for momentum conservation**

The viscous stresses in the momentum equation can be related to the rates of linear deformations of the fluid element. The rates of linear deformations are presented by the following formulae (Achkinadze et al, 2007):
- Linear elongating deformations:
\[ \varepsilon_x = \frac{\partial u}{\partial x}; \varepsilon_y = \frac{\partial v}{\partial y}; \varepsilon_z = \frac{\partial w}{\partial z} \]

- Linear shearing deformations:
\[ \gamma_x = \frac{1}{2} \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right); \gamma_y = \frac{1}{2} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right); \gamma_z = \frac{1}{2} \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \]

According to the Newton’s hypothesis, the viscous stresses are represented as follow:
\[ \tau_{xx} = 2\mu \frac{\partial u}{\partial x}; \tau_{yy} = 2\mu \frac{\partial v}{\partial y}; \tau_{zz} = 2\mu \frac{\partial w}{\partial z} \]
\[ \tau_{xy} = \tau_{yx} = \mu \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right); \tau_{xz} = \tau_{zx} = \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right); \tau_{yz} = \tau_{zy} = \mu \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \]

In the follow, we substitute the above expressions for viscous stresses into the momentum equations. For the x-, y- and z- momentum, one can write down:
\[ \rho \frac{D u}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( 2\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial v}{\partial x} + \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \right) \right] + S_{Mx} \]
\[ \rho \frac{D v}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( 2\mu \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial z} \left( \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \right) \right] + S_{My} \]
\[ \rho \frac{D w}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left( 2\mu \frac{\partial w}{\partial z} \right) + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial z} \right) + \frac{\partial}{\partial z} \left( \mu \left( \frac{\partial v}{\partial x} + \frac{\partial w}{\partial y} \right) \right) \right] + S_{Mz} \]

The equations are named Navier-Stokes equations, and they represent the form of momentum equations that is the most advantageous for the finite volume method formulation.

3.2. Reynolds equation for turbulence motion

According with theory, the Navier-Stokes Equations for Laminar Flow of Real Fluids represent the dynamic equilibrium equations between the inertial unitary forces on the one hand and the external unitary forces (e.g. mass, pressure and friction), on the other hand, applied to a fluid particle. This equation also applies to turbulent movements, only respect the condition that we have done the average of each term, so we can write:
\[ \frac{1}{T} \left[ \int_{t_0}^{T} \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) dt \right] = \frac{1}{T} \left[ \int_{t_0}^{T} \left( f_x - \frac{1}{\rho} \frac{\partial p}{\partial x} + v \Delta u \right) dt \right] \]

The group of terms representing the unitary convective force of inertia can be written as:
\[ u \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \]

By making the derivatives of the second member of equation, the term due the equation of continuity. Consider the fact that the mass forces are constant, and that the density is also constant (incompressible fluid) and the mediation rule applies. The equation becomes:
\[ \frac{\partial u}{\partial t} + \frac{\partial (u u)}{\partial x} + \frac{\partial (v u)}{\partial y} + \frac{\partial (w u)}{\partial z} = f_x - \frac{1}{\rho} \frac{\partial p}{\partial x} + v \Delta u \]
\[
\frac{\partial \tilde{u}}{\partial t} + \frac{\partial (\tilde{u} \tilde{u})}{\partial x} + \frac{\partial (\tilde{v} \tilde{u})}{\partial y} + \frac{\partial (\tilde{w} \tilde{u})}{\partial z} = f_x - \frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x} + \nu \Delta \tilde{u} - \frac{\partial (\tilde{u}' \tilde{u})}{\partial x} - \frac{\partial (\tilde{v}' \tilde{u})}{\partial y} - \frac{\partial (\tilde{w}' \tilde{u})}{\partial z}
\]

If we execute the derivation in the left part of the equation and considered the continuity equation \( \frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z} = 0 \), it follows:

\[
\frac{\partial \tilde{u}}{\partial t} + \frac{\partial \tilde{u}}{\partial x} + v \frac{\partial \tilde{u}}{\partial y} + w \frac{\partial \tilde{u}}{\partial z} = f_x - \frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x} + \nu \Delta \tilde{u} + \left[ \frac{\partial}{\partial x} (-\rho u' u') + \frac{\partial}{\partial y} (-\rho v' u') + \frac{\partial}{\partial z} (-\rho w' u') \right]
\]

The first term in the left-part of equation represents the local unitary force of inertia, the next three represents the unitary convective force of inertia. The first term in the right-part of equation is the mass unitary force, the second, the unitary pressure force, the third, the unitary viscosity force and the last, the unit forces due to the turbulent pulses.

Analogous equations are obtained in the directions Oy and Oz which together with the continuity equation represent Reynolds equations for turbulent motion.

3.3. Shear Stress Transport

Computational Fluid Dynamics (CFD) is a way of analyzing fluid flows numerically on a computer by a set of algebraic equations. The algebraic equations are obtained by discretizing the partial differential equations (PDEs), which may be momentum, conservation of mass, energy etc. The solution is attained at discrete points and hence the computational domain needs to be discretized into discrete areas or a volume - which is the mesh or grid. For this reason, the grid resolution is of great importance. For instance, in order to fully describe the physics of the flow close to a wall, the grid resolution need to be very precise and fine.

During the simulation, the Shear Stress Transport (SST) model in CFD was using software ANSYS. In the areas close to the surface of the ROV, the URANS equations will be used in conjunction with the k−ω SST (Shear Stress Transport) model of Menter. This model is basically a combination of the k−ω model by Wilcox and the standard k−ε model by Jones and Launder. Close to the cylinder surface, i.e. in the near-wall region, the original k−ω model is used. In the free shear layers and in the outer wake region, the standard k−ε model is used.

The k-x SST model is a common model for predicting the flow separation under adverse pressure gradient. The k-ω based SST model (Shear Stress Transport) accounts for the transport of the turbulent shear stress. Furthermore, the SST gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients (eddy-viscosity).

On the other hand, the BSL model combines the advantages of the Wilcoxs and the k-ε model. At the same time, still fails to properly predict the onset and amount of flow separation from smooth surfaces. The principal reason is that both models do not account for the transport of the turbulent shear stress. This is the reason that this results in an over-prediction of the eddy-viscosity. The proper transport behavior can be obtained from the eddy-viscosity:

\[
v_t = \frac{\alpha_1 k}{\max(a_1 \omega, SF_2)}
\]

Where \( v_t = \frac{\mu_t}{\rho} \)

\( F_2 \) is a blending function similar to \( F_1 \) function, the second function restricts the limiter to the wall boundary layer. During the underlying assumptions are not correct for free shear flows, S is an invariant measure of the strain rate.

We expected to have a turbulent movement in the fluid domain. In this case, the temperature is fixed at 25°C and the water is modeled and considered as an incompressible fluid. It is important to set
the fluid domain to be a large one, to analyze the damping force acting on ROV in CFD. The side, top, and bottom of fluid domain are modeled using free slip wall boundary conditions. The conditions used are presented in Table 2.

![Volumetric meshing for flow domain and close view of ROV in 3D view](image)

Figure 4. Volumetric meshing for flow domain and close view of ROV in 3D view

In this case, the physical state of the solution domain has been represented by a set of boundary conditions. The Default domain presents 2 boundary: the vehicle and the domain. The body vehicle present no slip wall and smooth wall roughness. The domain present the boundary type opening, a subsonic flow regime and for turbulence a Fractional Intensity 0.05 and Eddy Viscosity Ratio of 10.

![Momentum and Mass](image)

Figure 5. Momentum and Mass

Figure 5 shows the moments and mass of vehicle accumulated in 300 time step when the model was towed in the surge direction. We observe a decrease of value from 1.0e-025 to 1.0e-045 for the first 10 accumulated time step. After the first 10 step, the graphic present a constant value, around 1.0e-0.45. During the simulation, we see few curves variations of momentum and mass.
Figure 6. Turbulence

Figure 6 shows the convergence criteria for turbulence of vehicle. The value decrease for the first 12, respectively 50 steps, and then is constant.

Before performing the CFD, the mesh size needs to be defined properly as the shape of the boundary. It is recommended that the mesh size to be sufficiently small to capture the geometry of the ROV. For example, in this case, the number of elements represented approximately 756330 elements. The fluid domain has around 135403 nodes. The volume is 2232.9 m$^3$. The 3D view of volumetric mesh of the flow domain around ROV it is represented in Figure 4. In Table 3 are presented the extents of mesh.

| Extends       | Value               | Units |
|---------------|---------------------|-------|
| min x, max x  | -7.49959, 8.71459   | [m]   |
| min y, max y  | -8.10808, 8.10817   | [m]   |
| min z, max z  | -8.10813, 8.10831   | [m]   |

Figure 7.- Pressure of ROV and streamlines

In Figure 7 is presented the pressure around the vehicle. From figure, result a high-pressure region at the front of vehicle. As observed from simulation, there is certainly flow separation on the ROV. In Figure 7, the vehicle performs an immersion movement. As observed in Figure 7, the wakes
are formed at the rear and at the top of the ROV body when the turbulent/laminar flow past the vehicle. The minimum pressure has a value estimated at -4556.19 Pa situated at the bottom of the front, and a maxim pressure around 2709.41 Pa. When the vehicle performs a horizontal motion, reaches a minimum value of -3217.52 Pa, and a maximum value of 1952.23 Pa. We also observed the presence of streamlines. Those depend on the movement of vehicle: in horizontal plan or in vertical (immersion) plan. In Figure 7, the vehicle is in immersion plan.

In Figure 8 it is represented the graphic with the pressure distribution measured in equatorial plan of vehicle.

![Figure 8. Pressure distribution measured in equatorial plan of vehicle](image)

We observed an irregular distribution of pressure in equatorial plan of vehicle. For the first 0.1 m, the pressure suffer a decrease, after that the pressure increase reaching 1.500 Pa value. After the increase moment, the pressure decrease, reaching the most small value -1.900 Pa.

![Figure 9. Velocity of ROV and streamlines](image)

A maxim value reaches a value of 2.64 m/s, and a minimum value of 0.016 m/s. In Figure 9, the vehicle movement is in horizontal plan.

At the end of simulation, the software Ansys CFX perform the following information:
Table 4. Final results

| Variable Name                        | min    | max    |
|--------------------------------------|--------|--------|
| Density                              | 9.97E+02 | 9.97E+02 |
| Specific Heat Capacity at Constant Pressure | 4.18E+03 | 4.18E+03 |
| Dynamic Viscosity                    | 8.90E-04 | 8.90E-04 |
| Thermal Conductivity                 | 6.07E-01 | 6.07E-01 |
| Static Entropy                       | 0.00E+00 | 0.00E+00 |
| Velocity u                           | -1.36E+00 | 2.49E+00 |
| Velocity v                           | -1.99E+00 | 1.94E+00 |
| Velocity w                           | -1.82E+00 | 1.96E+00 |
| Pressure                             | -3.09E+03 | 2.00E+03 |
| Turbulence Kinetic Energy            | 5.29E-06 | 2.32E-01 |
| Turbulence Eddy Frequency            | 1.83E+00 | 1.62E+05 |
| Eddy Viscosity                      | 5.99E-08 | 1.24E+00 |
| Temperature                          | 2.98E+02 | 2.98E+02 |
| Wall Scale                           | 2.26E-04 | 5.31E+02 |
| Wall Distance                        | 1.32E-06 | 3.25E+01 |

Figure 10 The vehicle

The overall vehicle image up to now is shown in the Figure 10. Based on the shape design and arrangement of equipment on body, the design is made. In the future, the authors will examine and will test the vehicle for different speed and comparing the numerical simulation with the ROV.

4. Conclusion

In this paper was presented a systematic modeling of the hydrodynamic dragging forces of a remotely operated vehicle using Ansys CFX. 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] Robert D. Christ, Robert L. Wernli, Sr., The ROV Manual, A User Guide for Remotely Operated Vehicles, Second Edition;
[3] Thor I. Fossen, Models for Ships, Offshore Structures and Underwater Vehicles, 2011;
[4] Gianluca Antonelli, *Underwater Robots: Motion and Force Control of Vehicle-Manipulator Systems*, Springer Tracts in Advanced Robotics, New York, 2006.

[5] Fossen TI (1994) Guidance and control of ocean vehicles, Wiley, New York;

[6] Banerjee, A.K., Do, V.N., 1995. Deployment control of a cable connecting a ship to an underwater vehicle. Appl. Math. Comput. 70 (2), 97–116.

[7] Ohkusu, M., Koterayama, W., Kashiwagi, M., 1987. Hydrodynamics of a depth controlled towed vehicle. Oil and gas fields of Persian Gulf. Available from: ⟨http://meed.com/⟩; (accessed 11.08.15).

[8] Park, H., Jung, D., Koterayama, W., 2003. A numerical and experimental study on dynamics of a towed low tension cable. Appl. Ocean Res. 25 (5), 289–299.

[9] Boe, C., Rodriguez, J., Plazaola, C., Banfield, L., Fong, A., Caballero, R., Vega, A., 2013. A hydrodynamic assessment of a remotely operated underwater vehicle based on computational fluid dynamic–Part 1–numerical simulation. CMES: Comput. Model. Eng. Sci. 90 (2), 165–177.

[10] Fang, M.-C., Hou, C.-S., Luo, J.-H., 2007. On the motions of the underwater remotely operated vehicle with the umbilical cable effect. Ocean Eng. 34 (8), 1275–1289.

[11] Buckham, B., Nahon, M., Seto, M., Zhao, X., Lambert, C., 2003. Dynamics and control of a towed underwater vehicle system, part I: model development. Ocean Eng. 30 (4), 453–470.

[12] Kamali, R., Khojasteh, D., 2016. Investigation of dynamic behavior of a towed underwater vehicle at Persian Gulf. In: Proceedings of the 24th Annual International Conference on Mechanical Engineering-ISME, Yazd, Iran.

[13] Khojasteh, D., 2015. Dynamic Modeling of a Towed Undersea Body and Dynamic Study of a Point Absorber Wave Energy Converter (Master Thesis). Shiraz University, Shiraz, Iran.

[14] http://www.marine-knowledge.com/types_of_ships/an-underwater-robot/

[15] http://www.sciencedirect.com/

[16] SolidWorks software

[17] AnsysCFX software