

NUMERICAL RESEARCH ON THE UNDERWATER ROBOTS' HULL, IN A VIRTUAL WIND TUNNEL

Rus Simona, Diaconu Mihai, Zaharia Florin,¹
 Coșoiu Costin,² Ion Ana,³ Rus Victor - Octavian⁴.

¹ Diving Centre Constanta, simona_elenar@yahoo.com, ² Technical University of Civil Engineering Bucharest, ³ "Mircea cel Bătrân" – Naval Academy of Constanta, ⁴ Mechanics Faculty of the "Ovidius" University of Constanta, Romania.

Keywords: boundary layer, drag coefficient, virtual environment, wind tunnel, underwater robots' hull.

Abstract: This paper shows the results of numerical simulation of the flow around an underwater robot. The experiment was made in a virtual environment. The modeling was obtained as an efficient solution currently used worldwide in computational analysis of phenomena. The "FLUENT" software integrates numerically, in a given field, the momentum and energy conservation equations taking into account certain initial conditions on boundary contour area, making use of the continuity equation all through the iterative process. Numerical research aims mainly at validating the results of the experimental modeling.

1. INTRODUCTION

Numerical research on the shape of the underwater robots' hull, carried out in virtual wind tunnel aimed at validating the results of experimental modeling.

The FLUENT software fast meshing and uses formulas given in the 2 chapter on boundary layer theory.

Velocity distribution was presented in the close wake area of a virtual model on a vertical of the median longitudinal plane of the model, noting the existence of a speed deficit, whose parameters are closely related to the parameters of the submersible's drag coefficient. All these were observed along the axis of the model.

Research on drag coefficients for robot models with and without empennage (tail) and propulsion studied so far (i.e.: VSA, RSA and SM 358) are summarized in Table 1 in the 7 chapter.

2. THE EQUATIONS WITH WHICH THE "FLUENT" SOFTWARE OPERATES

The equations governing the uniqueness phenomenon and its conditions are:

a) the general equation of mass conservation, which is given by the formula:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \cdot \vec{v}) = S_m \quad (2.1)$$

where: ρ = the density of fluid,

v = the velocity of fluid,

S_m = the mass added to the continuous phase of the second dispersed phase,

∇ = operator with the following formula:

$$\nabla = \frac{\partial}{\partial x} \vec{i} + \frac{\partial}{\partial y} \vec{j} + \frac{\partial}{\partial z} \vec{k} \quad (2.2)$$

In general, the continuity equation for fluids, or the mass conservation equation is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \cdot v_x)}{\partial x} + \frac{\partial(\rho \cdot v_y)}{\partial y} + \frac{\partial(\rho \cdot v_z)}{\partial z} = 0 \quad (2.3)$$

where $\vec{v} = (\vec{v}_x, \vec{v}_y, \vec{v}_z)$ and $\frac{\partial \rho}{\partial t}$ disappear – because the phenomenon is permanent.

b) the equation of conservation of momentum (conservation of impulse equation) has the following formula:

$$\frac{\partial}{\partial t}(\rho \cdot \vec{v}) + \nabla \cdot (\rho \cdot \vec{v}\vec{v}) = \nabla p + \rho \cdot \vec{g} + \vec{F}_{ext} \quad (2.4)$$

where: pg = gravity,

F_{ext} = external forces acting on the fluid's surface bounded by the control surface,

p_s = static pressure; the other notations have the meanings mentioned above.

c) energy conservation equation with which the software operates is given by:

$$\frac{\partial}{\partial t}(\rho \cdot E) + \nabla \cdot (\vec{v}(\rho \cdot E + p)) = -\nabla \cdot \left(\sum_j h_j J_j \right) + S_h \quad (2.5)$$

The field calculation set before the problem is solved is bounded by the rigid surfaces of both the model, and the wind tunnel, as well as by the input / output air sections of the considered virtual tunnel.

3. INITIAL CONDITIONS AND IMPOSED LIMITS ON THE CONTOUR LINE BY THE SOFTWARE AND THE USER

In order study the movement of air around the model placed in a virtual tunnel the computation field should be set prior to problem solving, the domain is bordered by rigid surfaces of the model, of the tunnel, and input / output air sections. For the analyzed models there have been just one input, and one output. The limit condition for the input speed was set at:

$$\vec{v} = U_{\infty} = 22m/s \quad (3.1)$$

The initial conditions are specified either by the user or by other initial conditions that are specific for calculation algorithm, set by the computer software, which the user doesn't necessarily have to know.

The initial conditions we have attached to the problem were related to the speed of the moving fluid and by the pressure coming out of the field considered.

If there were no uniqueness conditions the problem wouldn't be completely defined, and thus, couldn't be solved. For example, if the speed at which water moves upstream weren't stated in advance, the computer wouldn't be able to display the distribution of speeds in the backwater.

On the other hand, the flow must be conserved. In other words, in a pipe the flow of water at both ends should be equal in quantity.

The model was placed in a virtual tunnel. Even so, the boundary conditions on the contour were defining it as a pipe with rectangular profile (conditions existing in the wind tunnel were simulated). Continuity equation must be respected. In the place where the model is fixed the flow section is reduced, while the flow remains constant; thus the velocity has to go up (see fig. 1), while close to the walls of the air current velocity is zero.

In conclusion, the boundary conditions are: $\vec{v} = 0$ on the contour (i. e. on the walls of the experimental system), the vein is in depression, \vec{v} is constant at the entry in the tunnel, flow is steady, and the pressure at the ends of the virtual tunnel is the atmospheric pressure.

4. RESULTS OBTAINED BY NUMERICAL MODELING

The figures below show the speed and pressure distributions in a median longitudinal plane which cuts the work surface, the pressure distributions on the surface of the model

and the velocity distribution in the wake on a vertical existing in a median longitudinal plane. By wake (backwater) we understand the track of turbulence left behind a ship, or any moving watercraft.

Figure 1 shows the distribution of local velocity rates in the vein of the virtual tunnel, and around the artificial VSA, while figure 2 shows the static pressure distribution in the vein of the virtual tunnel, and around the artificial model VSA. From the two figures an increase in the fluid's velocity can be observed, except in the boundary layer of the model's surface, and also a decrease of speed in its downstream as well as decrease in the static pressure around the model, and an increase upstream of the model.

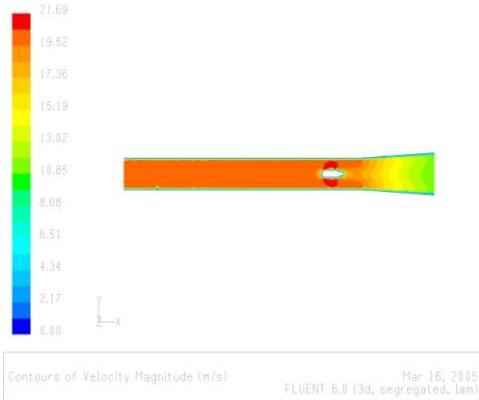


Figure 1: Distribution of velocity in the virtual tunnel virtual vein and around the virtual "VSA" model

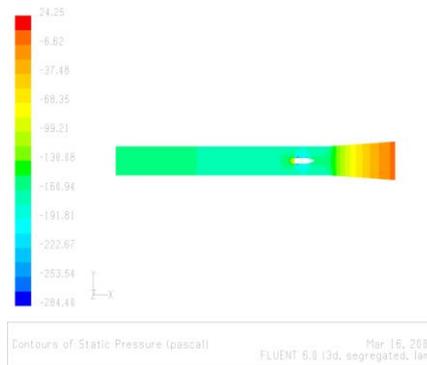


Figure 2: Distribution of static pressure in the virtual tunnel, and around the model

All through the virtual tunnel the flow is the same, and the velocity distribution on each body differs (fig. 1).

Comparing figures 1 and 2 one can observe that where velocity increases, on the other graph (figure 2) the static pressure decreases. The unit of local static pressure at a point on the surface of the model is shown in the chart devised by the FLUENT software in "Pascal"; the conversion will be as follows: 1 bar = 1 daN/cm² = 10⁵ N/m² (Pa) = 1.019 kgf/cm² (at) = 10.3 m col. H₂O = 14.5 psig. Figure 3 shows the distribution of local pressure on the lateral surface of the artificial model VSA. It highlights the high pressure on the bow area followed by a pressure decrease, to achieve then a minimum in the first third of the model, and ultimately an increase in pressure towards the backboard (stern) area of the model. It is worth noting that more than 90% of the lateral surface of the model is under pressure, which is true only for the model existing in the vein of the virtual tunnel. This phenomenon can be observed in figure 3.

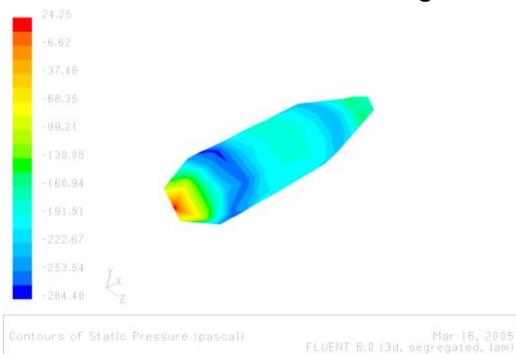


Figure 3: Distribution of local pressure area on the lateral surface of the "VSA" virtual model set with "FLUENT 6.0"

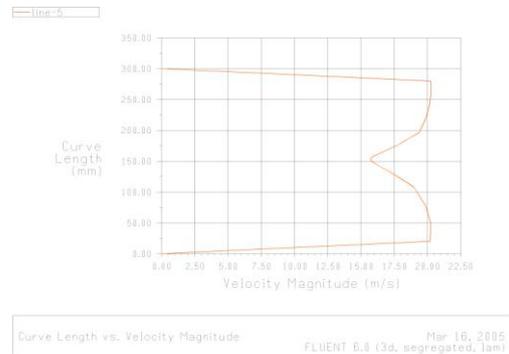


Figure 4: Distribution of velocity in the wake area on a vertical in the median longitudinal plane of the model

Figure 4 shows the distribution of velocity in the wake near the body of the VSA virtual model. There is zero speed on the wall of the virtual tunnel, as a boundary condition imposed by the adhesion property, and the existence of a velocity deficit right along the shaft of the model whose size is closely related to the size of the drag coefficient of the UV under observation.

The equation of continuity is used all along the virtual tunnel, i.e. the input flow is equal to the flow at any section along the tunnel. The FLUENT software meshes and uses the given formulas applying by the boundary layer theory. Attempts in the tunnel (hydro, wind) will be of particular importance because there is no thorough research in theoretical modeling (supposing to take into account all coefficients and factors involved along the way underwater robots cover). The results obtained in the virtual tunnel are compared with those obtained on a "TA 1 - LAIV" experimental tunnel model; conclusions will be drawn in the next section. They will be enlarged for the real robot observation.

5. ANALYSIS RESULTS OF THE MATHEMATICAL AND EXPERIMENTAL MODELING

Research on drag coefficients for three different forms of model submersibles with and without empennage and propulsion (VSA – autonomous underwater vehicle, RSA – self-propelled underwater robot, and SM 358 – diver-carrying submersible) are summarized in Table 1. The table shows: C_x coefficient values of the coefficients due to shape (distribution of local pressure on bodies' surface of the models investigated in virtual tunnel) $C_{x,p}$, due to friction along the boundary layer developed on the $C_{x,f}$ body surface, for determinations based on numerical modeling, and the values of the total C_x drag coefficients, for submersibles - with and without empennage and propulsion. These values can be obtained by physical modeling and by measurements on models placed in the experimental vein of "TA 1 - LAIV" wind tunnel.

Drag coefficients C_x (shown in Table 1) are established for Reynolds numbers (Re) provided in Table 1 ($Re_{VSA} = 8 \times 10^4$, $Re_{RSA} = 10^5$, $Re_{SM358} = 10^5$) and for the A_c amidships section shown in the same table (i.e. $A_{cVSA} = 2,83 \times 10^{-3} m^2$, $A_{cRSA} = 7,85 \times 10^{-3} m^2$ și $A_{cSM358} = 4,65 \times 10^{-3} m^2$).

6. FINAL CONSIDERATIONS

Numerical research presents the results of numerical simulation of the flow around an underwater robot hull. The place where the underwater robot is laid is a virtual representation of the "TA 1" wind tunnel located inside the Aerodynamic Laboratory "LAIV" of the Technical University of Civil Engineering Bucharest. A scale model of the submersible placed in the virtual tunnel aims at validating the results obtained after experiments are made.

At the beginning, the virtual model of the robot did not have propulsion and steering equipment.

The modeling was obtained by the help of the FLUENT software, version 6.0, for which the "LAIV" of the Department of Hydraulics and Environment is licensed. The software is installed on an up-to-the-minute computer. This is an efficient solution currently used worldwide in computational analysis of phenomena, which represents the subject of fluid dynamics (CFD - Computational Fluid Dynamics).

The software contains, numerically, the moment and energy conservation equations in a given area, taking into account certain initial conditions, ensuring that throughout the iterative process the equation of continuity is respected.

"FLUENT 6.0" determined: C_x , C_{xp} , and by difference there resulted the drag coefficient given by the hull's shape (C_{xf}) for underwater robots that have been observed so far.

7. DISCUSSIONS AND INTERPRETATION

In analyzing the results of the computer – assisted modeling (numerical and experimental) the following observations have been made in relation to the C_x drag coefficient values, determined theoretically and experimentally: the values of all coefficients are correspondingly higher for the propulsion and empennage robots; this phenomenon can be explained by the extension of lateral surfaces of the hulls, and by the disturbance of the flow in boundary layer around the hulls. This leads to a change in the distribution of local pressures on the lateral surface of the hull, and to a correspondent increase in the resistance of the C_x coefficients values.

In respect of the total C_x drag coefficients values, irrespective of the model type (with and without empennage and propulsion), the lowest values of the coefficients are established for the VSA model, while the highest ones are established for the RSA model; for the SM 358 model intermediate values are obtained. These differences arise from the different values of the drag coefficients of form (pressure), marked $C_{x,p}$, and are also due to different values of the drag (friction) coefficients $C_{x,f}$.

The analysis of values reveals that for the VSA model, characterized by a well-designed hull profile, coefficients $C_{x,p}$ and $C_{x,f}$ are the smallest if compared with the corresponding coefficients of the other models. For the RSA and SM 358 there is a substantial increase of $C_{x,f}$, due to the less convenient forms. The best shape in terms of drag is that of the VSA virtual model.

These coefficients can be theoretically and experimentally approximated, due to the rigorous compliance with the conditions of specific similitude, as having the same value on both model and prototype. It appears obvious that in order to determine the hydrodynamic drag forces for all submerged types studied, the same number Re for which modeling was done the speed values of real submersibles are used; the following values are also taken into consideration: characteristic lengths, amidships sections, densities of aquatic environment in which they work.

Table 1 C_x , $C_{x,p}$ and $C_{x,f}$ drag coefficient value obtained as a result of numerical and experimental modeling for the robots under observation

R o b o t s	Reynolds numbers Re [-]	A_c [m ²]	Drag coefficient value " C_x " obtained as a result of numerical modeling						" C_x " value obtained as a result of experimental modeling	
			Robots without empennage and propulsion in virtual tunnel			Robots with empennage and propulsion in virtual tunnel			Robots without tail and propulsion in the experimental "TA 1" tunnel	Robots with tail and propulsion in the experimental "TA 1" tunnel
			$C_{x,p}$	$C_{x,f}$	C_x	$C_{x,p}$	$C_{x,f}$	C_x	C_x	C_x
1	2	3	4	5	6	7	8	9	10	11
VSA	$8 \cdot 10^4$	$2,83 \cdot 10^{-3}$	0,456	0,004	0,460	0,465	0,006	0,471	0,485	0,506
RSA	10^5	$7,85 \cdot 10^{-3}$	0,520	0,030	0,550	0,534	0,032	0,566	0,571	0,590
SM 358	10^5	$4,65 \cdot 10^{-3}$	0,488	0,042	0,530	0,488	0,044	0,532	0,534	0,536

8. CONCLUSIONS

This paper presents a theoretical study on finding the optimal shape of a hull for

underwater research robot so that the drag and the energy consumption should be kept to a minimum (energy consumption is one of the major problems of AUR) while the robot cuts through its work site. Thus, the number of the robot's working hours is expected to grow.

The tests in hydro and wind tunnels will be of particular importance because there has not been any thorough theoretical modeling, so far (especially for those which are supposed to take into account all coefficients and factors involved in underwater robots crossing). The results obtained in virtual tunnel are to be compared with those obtained on a TA 1 tunnel models, and conclusions drawn will be extended and applied to the real robot.

By analyzing the " C_x " coefficients determined theoretically and experimentally we deduce that, for the three models, the values of all coefficients are higher for the robots with propulsion and empennage (tail) than for the similar ones without propulsion and empennage. This is due to the extension of lateral surfaces of the bodies and to the disturbance of the flow all around the hulls (noticeable when visualized).

Tests have shown that for the RSA, and SM 358 models there is a change in the local distribution of pressure on the lateral surface of the hull, a substantial increase in $C_{x,f}$ friction coefficients, and an implicit increase in the values of the total drag coefficients " C_x ".

Coefficients " C_x " have the lowest values in the VSA model, which has a better shaped hull. In this case the values of $C_{x,p}$ pressure coefficients and $C_{x,f}$ friction coefficients have the lowest value if compared to those of other models.

So, of the three forms studied, the best in terms of drag is the model chosen for the VSA - autonomous underwater vehicle.

In conclusion the paper presents the numerical research using the FLUENT software which operates on the basis of boundary layer theory.

The software integrates numerically, in a given field, the momentum equations and energy conservation equations, indicated in the second chapter, taking into account certain initial conditions on boundary contour area, making use of the continuity equation all through the iterative process. Numerical research aims mainly at validating the results of the experimental modeling.

References:

1. MD Ship Hydromechanics Dept – Investigation of the stability and control characteristics of several configurations of the DARPA suboff model (DTRC model 5470) from captive – model experiments – US National Technical Information Service, 1990.
2. Degeratu M., Georgescu A., Hașegan L. i.e., Dynamic wind tunnel tests for the Bucharest Tower Center, Scientific Bulletin of the Politehnica University of Timișoara, Transactions on Mechanics, Tom 51(65) Fascicola 3, Special Issue ISSN 1224-6077, pp.84-88, Timișoara, 2006.
3. Barlow J. B., Rae W. H., Pope A., Low-speed wind tunnel testing, John Wiley & Sons, New York, 1999.
4. Submarine seamanship – Defense Ministry – Divers Centre Constanta, 1989.