23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

DRAG FORCE EVALUATION OF A REMOTELY OPERATED VEHICLE

José Bismark de Medeiros[1], jose.bismark@univasf.edu.br
Lucas Aninger de Barros Rocha[2], lucas.aninger@gmail.com
Luiz Mariano Pereira[3], luiz.mariano@univasf.edu.br
[1,2,3] Universidade Federal do Vale do São Francisco, Av. Antonio Carlos Magalhães, 510 – Santo Antônio, CEP.: 48.902-300, Juazeiro – Bahia, Brasil.

Abstract. *In Brazil, operations such as maintenance and deepwater exploration have been growing for years. However, much of the equipment for these procedures is still mostly foreign. For the development of national technologies in submerged vehicles, a lot needs to be done, especially in the analysis of drag forces caused by ocean currents, one of the most important variables in the design of this type of machinery. Therefore, this work aims to determine the drag force on submerged Remotely Operated Vehicle (ROV) using Computational Fluid Dynamics (CFD), in order to assist the proper sizing and insurance propulsion systems required for its operation. To analyze this, various computer simulations were performed using ANSYS software for determining the drag coefficient of the ROV. Several geometries were studied to obtain results that could be easily compared with results available in the literature for basic geometries, as well as for the purpose of determining the reliability of more complex further analysis. Finally, the geometry simulation of a three-dimensional commercial vehicle was modeled using SolidWorks software. The option to analyze an actual vehicle was considered with the goal of obtaining qualitative information on the drag coefficient. The manufacturer and a company operating this kind of vehicle in Brazil have shown interest in conducting experimental tests to compare with those obtained computationally. This work presents results for drag coefficient obtained up to the operating velocity range of the ROV provided by a manufacturer.*

Keywords: *ROV, CFD, Fluid Mechanics; Computational Modeling.*

1. INTRODUCTION

This work is a part of the initiative of submersible vehicle construction, for shallow, medium and deep depths, named Dragon of the Sea – Development of the Large Aquatic Robot, Advanced, Offshore, Dedicated to Operation, Maintenance, Auxiliary and Repair, with financial support in the range of the CHAMADA PÚBLICA MCT/FINEP/AT – Cooperation of ICTs – Businesses – PRÉ-SAL – 3/2010. Sponsors of the project include BWV Consultoria and the ARMTEC Tecnologia em Robótica, of the state of Ceará, both in partnership with the Center of Information Technology Renato Archer (CTI), of the Federal Government, with the Federal University of the Valley of the São Francisco – UNIVASF as a participant of the Scientific and Technological Institute (ICT).

For Madeira and Souza (2012), the ROVs (Remotely Operated Vehicles) are vehicles of great importance due to the possibility of being produced in diverse sizes and capacities for achieving activities with efficiency in places where human support it is not possible due to pressure, reaching or passing the time necessary for operation. Figure 1 represents a schematic model of the composition of an ROV, the umbilical cable, and the control center.

In the project of subaquatic vehicles, the drag force is important due to various factors that it produces: the friction between the blades and between the fluid and the body, and the relative speed between the fluid and wet parts with running currents. Among the critical points in the study of ROVs is the estimation of the forces that influence the structural sizing in conjunction with motorized propellers of these vehicles, whose power is a function of the drag force as well; this is the object of study in this work. The structural project should focus on the meeting the requirements of resistance to high pressures, both static and dynamic, and protection of the equipment in use, in addition to reducing the volume and facilitating emersion and submersion, among other requirements (Magalhães, 2007 e Nomoto e Hattori, 1986). To focus on all of these premises, geometries in the form of a cage are generally adopted, despite the fact that they produce large drag efforts when the vehicle is moving, which decreases their functional performance. Because of this, this work searches for a determination of the drag force in submersed vehicles operated remotely (ROV), utilizing Computational Fluid Dynamics (CFD), with the intention of aiding the adequate and secure sizing of the propulsion system necessary for its operation.

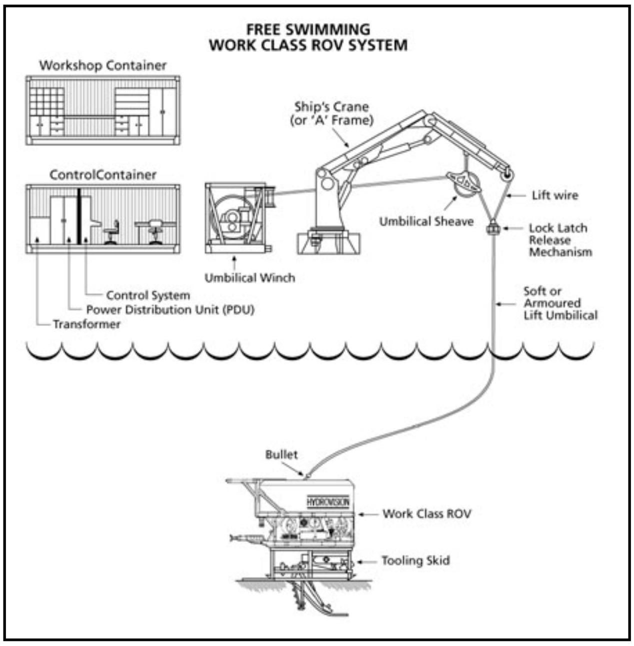


Figure 1. Schematic example of a ROV. Source: Ferreira, 2006.

2. ANALYSIS OF THE DRAG FORCES

According to (Fox, 2010), a viscous fluid flowing around a body generates superficial tensions around it, which have shearing components (tangential) and pressure components (normal), due to the viscosity and the local pressure, respectively. Mathematically, the force that generates these tensions can be described by Eqs. (1-3):

$$\vec{F} = \int_{\text{body surface}} d\vec{F} = \int_{\text{body surface}} d\vec{F}_{\text{shear}} + \int_{\text{body surface}} d\vec{F}_{\text{pressure}} \quad (1)$$

where,

$$d\vec{F}_{\text{shear}} = \vec{\tau}_{\omega}\, dA \quad (2)$$

and,

$$d\vec{F}_{\text{pressure}} = -P d\vec{A} \quad (3)$$

The application of these Equations (1-3) appears simple with a prospect of obtaining analytical solutions. However, after working on these expressions and using the Pi theory of Buckingham, we arrive at a dimensionless fluid dynamic coefficient of drag, C_d, depending only on the Reynolds number, *Re*, which is:

$$C_d = f(Re) \quad (4)$$

Knowing the Reynolds number and the geometry of the body, we can calculate the distribution of pressure, after integrating the surface pressure, thus meeting the liquid force arising from the pressure components. Now, utilizing the distribution of pressure, determining the superficial viscous tension, and integrating the surface tension again, we will have the resulting force. Nevertheless, this procedure is much more complex than it appears to be, as it is only possible for bodies with simple geometries, because beyond the difficulty of the operations for complex geometries, it generates erroneous results when the detachment of the boundary layer and its consequences for runoff are taken into consideration. Thus, the analysis of the drag forces is complete, predominately, for the experimental and numerical/computational methods called CFD, or Computational Fluid Dynamics. In this work, a CFD module of the commercial software ANSYS® was used for simulation and calculations of the drag forces. The mathematical models for the diverse types of steady flow, using the CFD software, are based on the equations of conservation of mass, the quantity of movement (Navier-Stokes), and energy. It seeks, therefore, as main objective to obtain the active drag forces on a ROV vehicle model, determining the contribution of these forces around the sizing of the propulsion systems.

3. MATERIALS AND METHODS

In order to execute a CFD analysis, it is necessary to define the set of equations and the region, called the domain, in which these are valid (Fortuna, 2000). In face of the difficulties in obtaining numerical solutions for continuous domain, due to the presence of infinite points, this physical domain should be discretized into a finite set of points, which results in a computational domain so called mesh, from which the solutions will be generated. The better distribution of these points, that they are refined to regions of interest, the larger the reliability of the results of the simulation. After the generation of the refined mesh, the dominant written equations, by the software, function as unknown values attained. This set of equations is joined to additional boundary conditions and to the initial conditions as properties of fluid and configuration of flow. Finally, the equations are solved after determining a number of iterations and attending to the stopping standards, proving the result to be analyzed and verified, the meaning of the results, to validate the solution in some form as compared to experimental results. The verification must be objective analysis if the mathematical model of the problem results in consistent solutions, and at that point it is reliable.

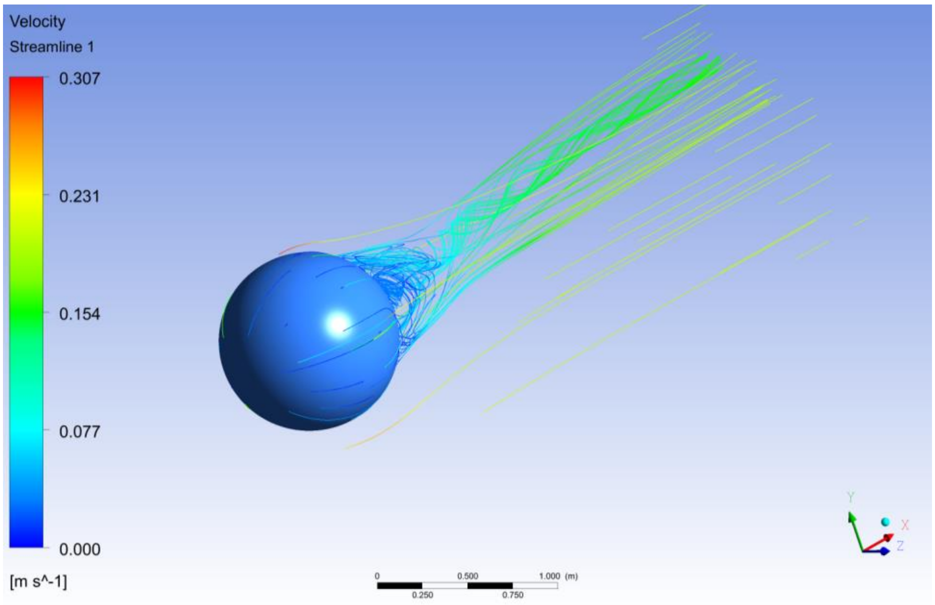
Having an understanding of this necessity, because there are not experimental data of the ROV model dealt with here, the validation of the mathematical model was completely simulated with experimental results combined to the literature, for example those presented in (Munson et al., 2004). The authors present diverse basic geometrical forms, a dimensional parameter (normally a ratio between length and diameter), the Reynolds number and its respective drag coefficients, as described in Tab. 1.

Table 1. Approximate values of the drag coefficient for bodies of diverse forms

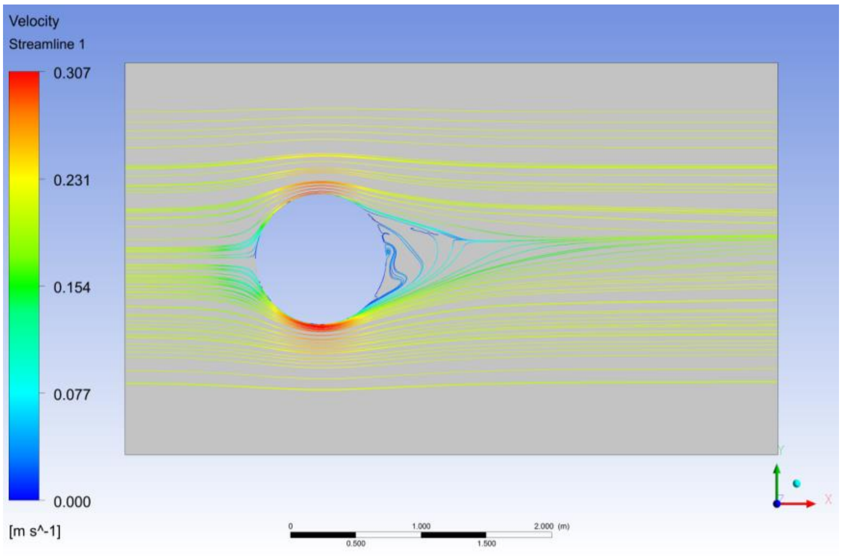
Shape of obstruction	L/D	Re	Cd
Circular cylinder with diameter D, length L and parallel axis to the fluid current	2	$> 10^3$	0.85
Circular cylinder with diameter D, length L and perpendicular axis to the fluid current	5	10^5	0.74
Sphere	-	10^5	0.50

Source: Assy, 2004.

Initially, the flow around the sphere with a diameter of 0.5 m was studied, with a Reynolds number of 10^5, and the streamlines of velocity were showing in Figs. 2. The 3D model of the sphere was obtained through the software SolidWorks®. The sphere was inserted into a rectangular cubical domain of 27 m^3 (3m x 3m x 3m). The model was imported into the platform ANSYS®, utilizing the analysis module of Computational Fluid Dynamics FLUENT®. The mesh was generated and refined, as can be observed in Figs. 3.

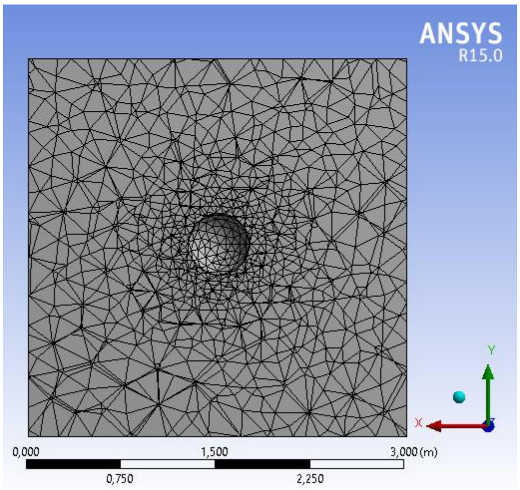


(a)

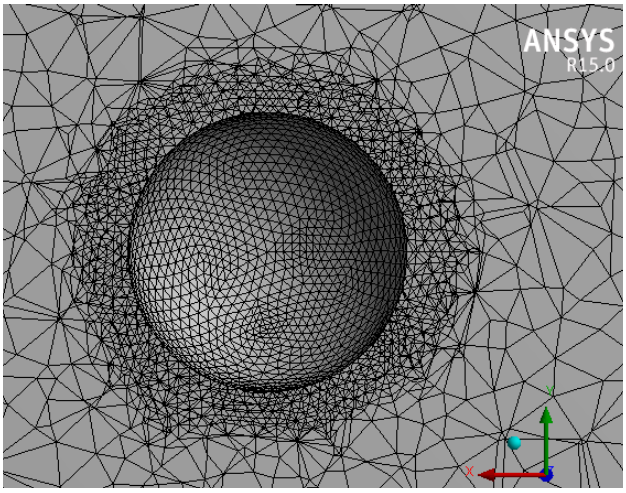


(b)

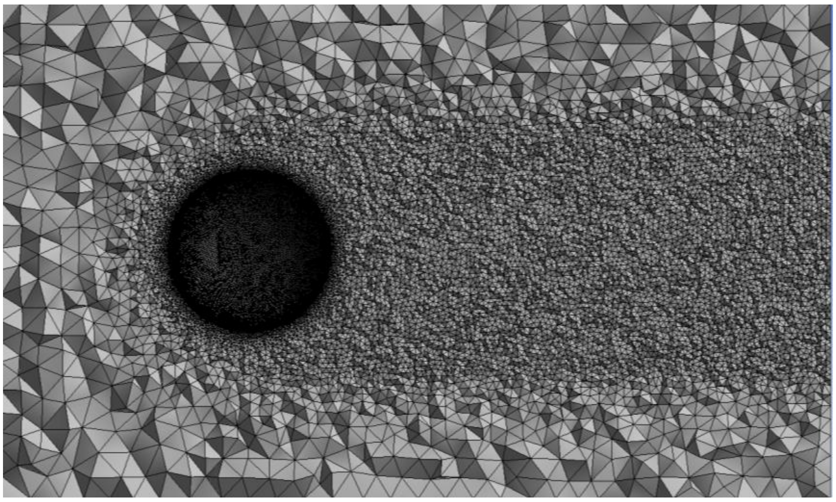
Figure 2. Simulation of flow around a sphere. (a) 3D velocity streamlines (b) 2D velocity streamlines
Source: Authors of this paper.



(a)



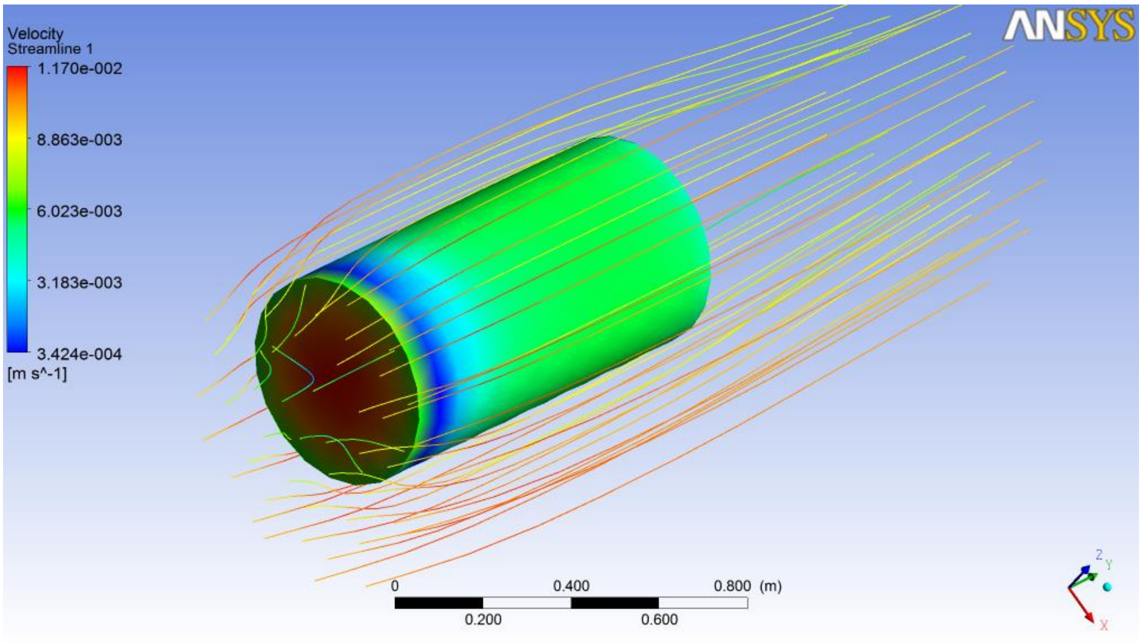
(b)



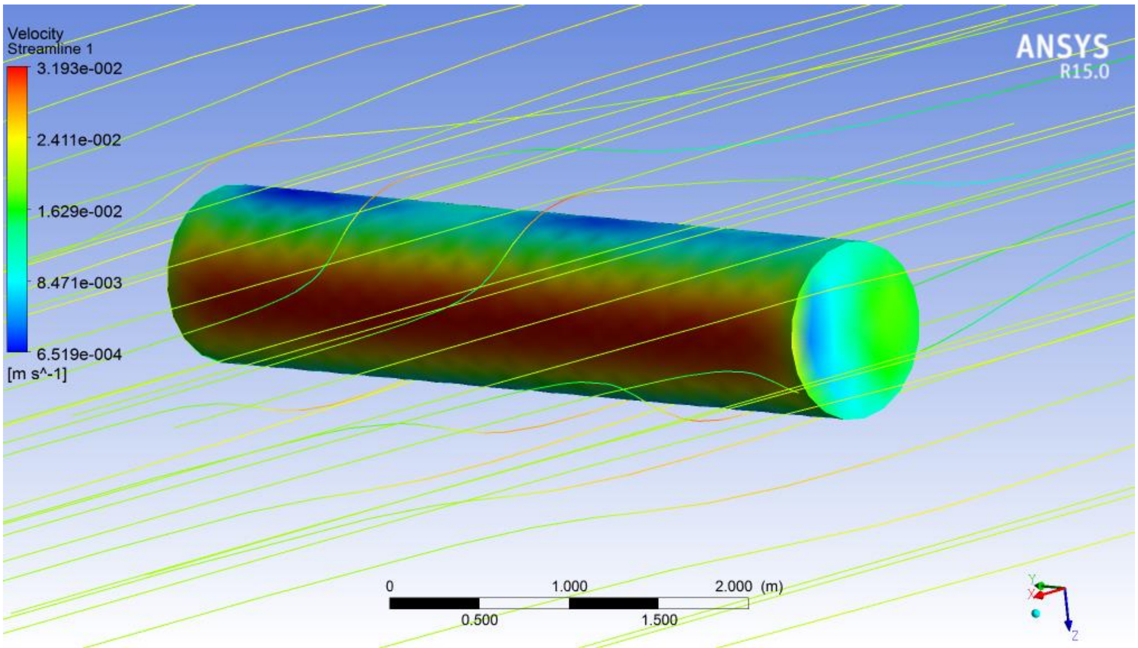
(c)

Figure 3. View of transverse section of the domain with mesh generated (a) automatically, (b) refined around the sphere and (c) refined along the wake. Source: Authors of this paper.

With the mesh configured, the boundary conditions were applied to the control surfaces, in addition to informing the value of the velocity of the flow and of the kinematic viscosity *v*. The work fluid, in this case, was water, without considering the heat transfer effects. The turbulence model adopted was the k-ε model as a better fitting configuration of flow in this simulation. The region of entry and exit of the flow, the surface of the sphere as a solid obstacle, and the control parameters of the solution as a minimum and maximum number of iterations were all defined. The covergence criteria that, for this simulation, is indicated by the remainder of calculation (the difference between iterations) for half of the evaluations of the quadratic remainder average (RMS – Root Mean Square), defined as the criterion of stopping, for this example, in 10^{-6}, thus establish a good level of convergence for the executed simulations. The same prodecure was repeated for two more flow situations around different geometries, as seen in Figs 4 (a) and (b).



(a)



(b)

Figure 4. Cylinder with flow (a) parallel (b) perpendicular to the longitudinal axis. Source: Authors of this paper.

The results and comparison with the specialized literature are described in Tab. 2.

Table 2. Comparison of the actual results for the drag coefficients with the reference values.

Shape of the Obstacle	Assy, 2004	Cd obtained in simulation	Deviation
Sphere	0.50	0.4499	10.02%
Circular cylinder with diameter D, length L and axis parallel to the fluid current	0.85	0.9113	7.21%
Circular cylinder with diameter D, length L and axis perpendicular to the fluid current	0.74	0.8293	12.07%

4. RESULTS

Faced with the additional necessity of comparing the results obtained in these simulations as they are developed for a more complex geometry, the submarine vehicle chosen for this simulation was the SeaBotix LBV200-4, which was indicated by the partners of this research, ARMTEC Robotic Technology and the CTI – the Renato Archer Information Technology Center, united in the research with the Ministry of Science, Information, and Technology (MCTI). Thus, after the future comparisons of the results obtained in these simulations with experimental results, there will be greater confidence of the estimation of the drag force for the configuration to be chosen.

With the geometrical specifications and the existing operation of the ROV, a 3D model was constructed with SolidWorks®, Fig. 5. In the model presented in this figure, a simplified geometry of the model exists when compared to the real vehicle. However, such simplification was completed with the purpose of diminishing the complexities and details that would lessen the impact of the computational processing capacity available.

(a) (b)

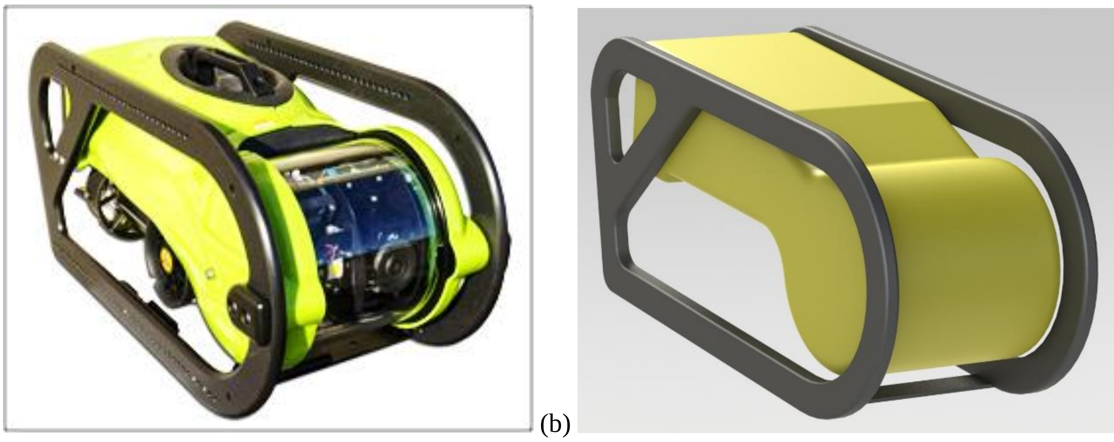


Figure 5. SEABOTIX LBV200-4. Real image (a) and computational modeling (b).
Source: a) SEABOTIX, 2012. b) The authors of this paper.

Following the steps utilized in the evaluation of the drag forces around the model of the vehicle selected, after the generation of mesh and appropriate refinements, according to Fig. 6, the simulations were accomplished for their own speeds of operation of the real vehicle, for example, 1.02 m/s, the maximum speed of the current indicated in the operation manual.

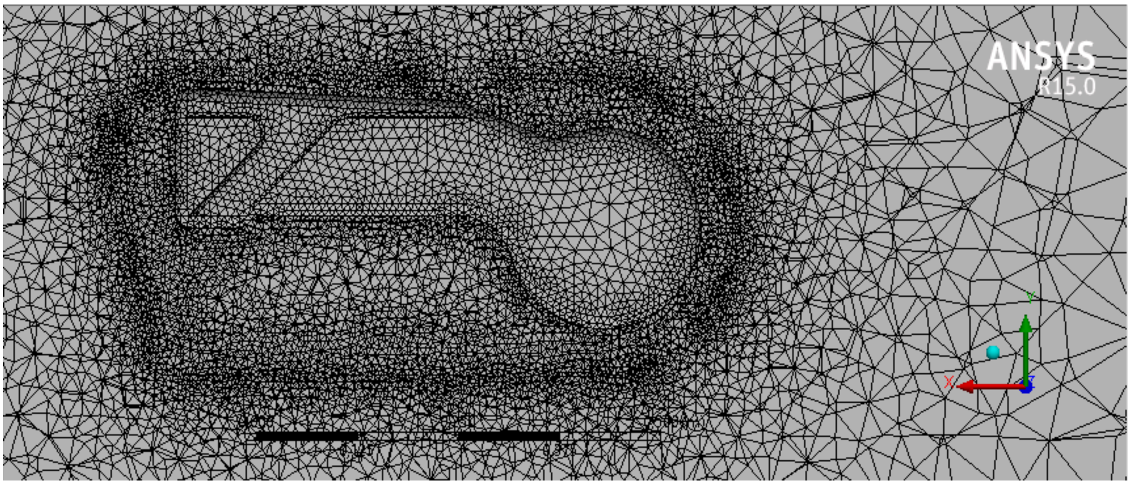


Figure 6. Mesh refined for simulation of the selected ROV. Source: The authors of this paper.

All of the necessary parameters were configured for the simulation, obtaining a drag force of 22.19N, and using the designed front of the ROV, the drag coefficient was calculated to be 0.4723. Figure 7 demonstrates the performance of the fields of pressure in the flow around the vehicle.

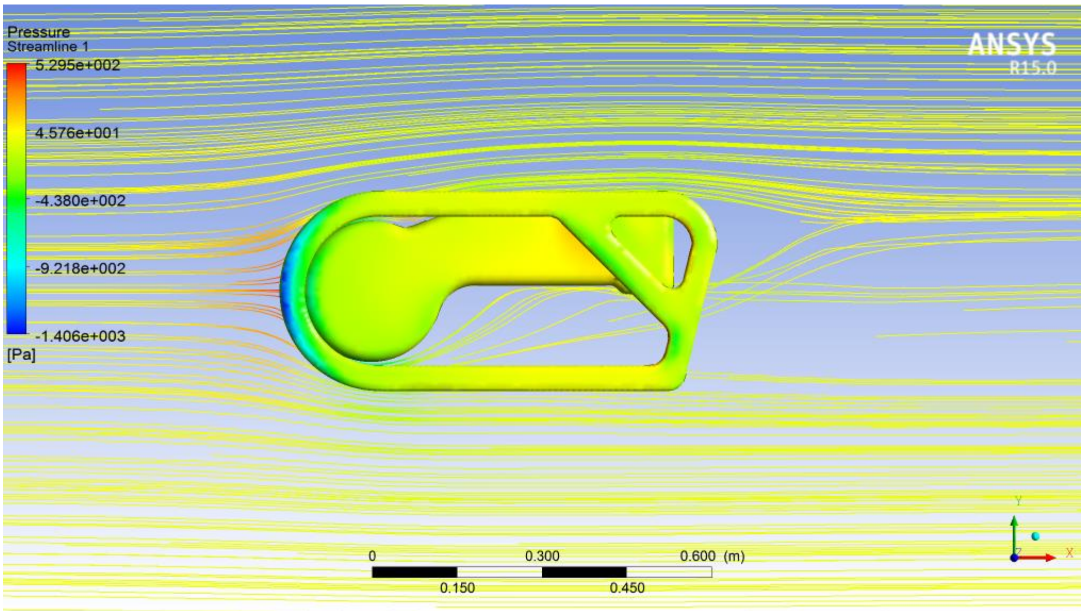


Figure 7. Simulation of runoff around the LBV200-4. Level curve of the pressure field.
Source: The authors of this paper.

5. DISCUSSION

The principle objective of this research was to help the company ARMTEC Robotic Technology and the CTI in the development of a remotely operated submarine vehicle, ROV, with national technologies, supporting the improvement and domain of computational techniques of simulation used in the problems of fluid flow around bodies. With this work it was possible to estimate of the capacity of thrust that propulsion systems of a subaquatic vehicle should be capable of coping with. However, note that the purpose here was to present a sequence of necessary steps for the development of a more complex simulation, currently in development, but the simplified models are necessary for comparison and evaluation of the reliability of the results.

6. ACKNOWLEDGMENTS

The authors thank the Foundation of Support for Research in the State of Bahia – FAPESB, for the initial scientific funding of the student Lucas Aninger de Barros Rocha, the contribution and support of the company ARMTEC Robotic Technology, of CTI and of the Financer of Studies and Projects – FINEP.

7. REFERENCES

ARMTEC, 2015. Tecnologia Robótica. <http://v2.armtecbrasil.com/index.php?parceiros>.

Assy, T. M., 2004. Mecânica dos Fluidos – Fundamentos e Aplicações. Rio de Janeiro, LTC.

CTI, 2015. Centro de Tecnologia da Informação Renato Archner. <http://www.cti.gov.br/>.

Fortuna, A. O., 2000. Técnicas Computacionais para Diâmica dos Fluídos. São Paulo. Editora da Universidade de São Paulo.

Fox, R. W. and McDonald, A. T., 2010. Introdução à mecânica dos fluidos. Rio de Janeiro: LTC.

Madeira, C. H.; Souza, N. C. A. *Projeto "Podeison": Veículo subaquático operado remotamente. 2012*. Trabalho de Conclusão de Curso (Graduação em Engenharia Mecânica) – Universidade de Brasília, Brasília, 2012.

Magalhães, P. H. V., 2007. *Desenvolvimento de um submersível remotamente operado de baixo custo e caracterização dos sistemas de propulsão e vetorização de empuxo por hélice*. Belo Horizonte, 2007. Tese de doutorado – Universidade Federal de Minas Gerais.

Munson, B. R.; Young, D. F. and Okiishi, T.H., 2004. Fundamentos da Mecânica dos Fluidos. São Paulo. Edgard Blücher.

Nomoto, M. and Hattori, M, 1986. "A Deep ROV "DOLPHIN 3K": Design and Performance Analysis". *IEEE Journal of Oceanic Engineering*, Vol. OE-11, No. 3, 1986;

SEABOTIX. LBV200-4: Vehicle Specifications. San Diego/CA, EUA, 2012. Disponível em: <http://www.seabotix.com/products/lbv200-4.htm>. Acesso em: 08 set. 2013.

8. RESPONSIBILITY NOTICE

The authors are the only people responsible for the printed material included in this paper.