ENC-2020-0513
Prediction of Drag on Submarines Using OpenFOAM

Filipe Dutra da Silva
Eduardo de Bittencourt Ribeiro
Thiago Pontin Tancredi
Ernane Silva
TEG - Thermofluids Engineering Group, Federal University of Santa Catarina, Joinville-SC, 89219-600, Brazil
filipe.dutra@teg.ufsc.br, eduardo.ribeiro@teg.ufsc.br, thiago.tancredi@teg.ufsc.br, ernane.silva@teg.ufsc.br

Abstract. A simulation model to estimate drag of an axisymmetric submarine hull was built using the open source code OpenFOAM. The objective is to obtain modeling parameters which represent, with efficiency, the movement of such vessels. Numerical errors due to grid refinement and sensitivity to boundary conditions were assessed and validation through comparisons with experimental data is provided. The analyzed Reynolds numbers ranged from 4×10^6 to 2.5×10^7. Results showed good agreement with experimental data. The low computational cost and the simple mesh generating process makes the simulation model suitable for use in optimization routines and analyses of complex geometries. Such simulations can also allow preliminary estimates for the design of the propulsive and maneuvering systems of submarines.

Keywords: Drag computation, Hull design, Submarine, OpenFOAM

1. INTRODUCTION

During the last decades, numerical methods have been used for the performance analysis of complex engineering systems and its use has become more and more usual during the preliminary design stages of any product. However, the application of such techniques is still limited in the area of Naval Architecture and Marine Engineering, possibly due to the empirical character of this field or the lack of data in the literature to validate the numerical models. This is more evident for submarine design, where the use of Computational Fluid Dynamics (CFD) techniques is restricted basically to military applications. Many studies in the literature have proposed hull geometries for submarines aiming at improving the internal design and reducing the drag. These studies were of paramount importance to reach the hull geometries currently used.

Landweber and Gertler (1950) and Jackson (1992) made important contributions to submarine design proposing the teardrop-shaped hull, which in theory presents the best compromise between displacement and drag. Both studies presented close results depending on the empirical coefficients adopted, though different expressions were used to describe the hull geometry. More recently, Joubert (2004, 2006) shedded some light on submarine hydrodynamics, proposing a hull with elliptical nose and parabolic tail, similar to that proposed by Jackson (1992). However, Joubert (2004, 2006) did not present the expressions used to describe the hull geometry.

Gertler (1950) reported the first systematic measurements of drag for hulls developed from forms of series 58, which were proposed earlier by Landweber and Gertler (1950). The authors showed that the drag at deep submergence is affected by five geometric parameters: fineness ratio, prismatic coefficient, nose radius, tail radius and position of the maximum section. The results indicated that drag varies considerably with the hull geometry and a minimum resistance can be achieved by tuning each geometric parameter individually, except the nose radius. This study supported the design of the USS Albacore, the forerunner of all well-succeed submarines from the US Navy Landweber and Gertler (1950).

Thereafter, many other studies have reported experimental data of drag at deep submergence for bodies of revolution, such as Larsen (1955), Randwijck and Feldma (2000) and Fureby *et al.* (2016), which still find large applicability for the preliminary design of submarines. Liu and Huang (1998) also presented experimental data for the DARPA Suboff submarine, which are widely used nowadays in conjunction with the ONR Body1 and the DTRC 4621 to validate numerical models using CFD. The high cost of experimental measurements have bolstered the applicability of CFD in the development of systematic series, which serve as the baseline for the design of vessels.

The Reynolds-averaged Navier–Stokes (RANS) equations are widely used in CFD simulations to predict the drag at deep submergence for bodies of revolution. Mulvihill and Yang (2007), for instance, applied a RANS model to evaluate the hydrodynamic effect of the individual components of a ONR Body1 submarine and the resulting force and momentum coefficients. Pan *et al.* (2012) predicted the drag coefficient during the heave and pitch motions of a DARPA Suboff submarine using RANS. Zhang *et al.* (2013) and Holloway *et al.* (2015) performed RANS simulations to obtain the force and momentum coefficients for DRDC STR, DARPA Suboff and DRTC 46621 submarines. The same authors have also

evaluated the flow separation under constant speed for all geometries considered. Kouh *et al.* (2009) used a RANS model to investigate the effect of scaling on the shape factor of a DTRC 4621 submarine. More recently, Fureby *et al.* (2016) investigated the flow over a DSTO submarine with appendices under different yaw angles. These authors showed that the RANS model predicts the pressure coefficient accurately when compared to experimental data but fails to predict the velocity fluctuations near the hull. To overcome this limitation of the model, they suggested the use of direct numerical simulation (DNS) or hybrid approaches, even though the computational costs are expected to increase significantly.

This study aims to develop a CFD model using the open source code OpenFOAM (The OpenFOAM Foundation, 2018) to predict the drag on axisymmetric hulls for submarines. Particularly, the flow over a model of series 58 is investigated and the numerical predictions are compared to experimental data provided by Gertler (1950). In section 2, information of the reference geometry as well as details of the simulation model can be found. Section 3 presents the main results and discussion. Finally, the overall conclusions are presented in section 4.

2. METHODS

2.1 Geometry and Experimental Data

The chosen geometry is the hull model 4165 from series 58, studied by Gertler (1950). The main geometric characteristics of this hull model, depicted in Fig. 1, are length $L = 2.74$m, wetted area $S = 2.491m^2$ and maximum diameter $D = 0.392$m. The experiments were performed by Gertler (1950) with the submerged hulls at different speeds. In some of the tests, the authors made use of a sand strip near the leading edge to ensure a fully turbulent condition in the boundary layer. For the purpose of validation of the numerical results, the total drag coefficient data (c_t) from Gertler (1950) was corrected by subtracting the influence of the sand strip and towing struts.
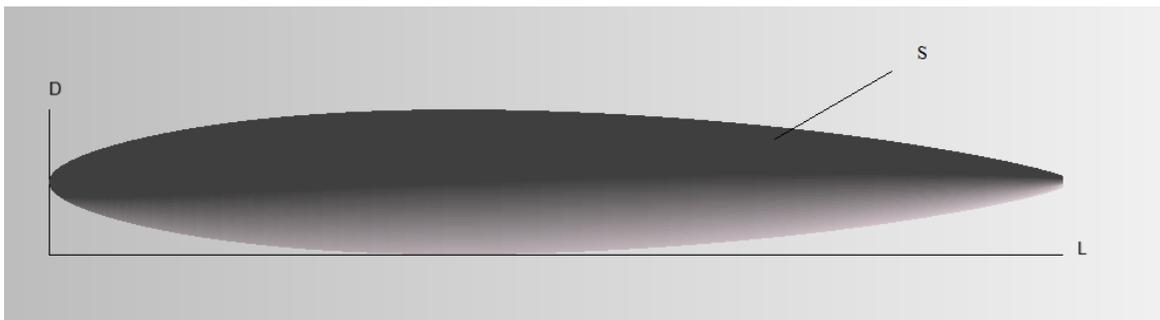

Figure 1: Modeled geometry of hull 4165 from series 58 Gertler (1950).

2.2 Simulation model

In the present study, incompressible Reynolds Averaged Navier-Stokes (RANS) simulations were conducted using the SST turbulence model (Menter, 1994) implemented in OpenFOAM v6.0. OpenFOAM is an open source CFD library with numerical solvers for a variety of physical problems, which is widely used by academy and industry. The computational domain and boundary conditions are depicted in Fig. 2. In order to test the sensitivity of the solution to the domain size, three domains were considered with dimensions described in Tab. 1.

The computational grid was generated using the SnappyHexMesh tool (The OpenFOAM Foundation, 2018), and is shown in Fig. 3. The grid is formed predominantly by cubic volumes, which are organized in seven refinement regions. The grid size increases by a factor of two from the inner refinement regions to their outer adjacent ones. Close to the hull, the cubic volumes are trimmed in order to respect the surface geometry. Layers of prismatic volumes are added near walls to allow a suitable discretization of the boundary layer gradients. The height of the first volume and number of layers vary with grid refinement, while the thickness of the complete layer region is kept constant for all the grids. The analysis of grid refinement is presented in section 3.

A freestream boundary condition was imposed in the frontal and lateral sides of the domain. The freestream velocity was set in accordance with the target Reynolds number. A prescribed freestream value of 0 Pa was imposed for gauge pressure. Regarding turbulence quantities, values of k and ω were prescribed based on a turbulence intensity of $I = 0.5\%$ and eddy viscosity ratio $\nu_t/\nu = 10$. Sensitivity tests for the turbulence variables are presented in section 3.

For the outflow condition, a prescribed back pressure value of 0 was imposed and a zero gradient condition was used for the other properties. On the surface of the model, a no-slip boundary condition was imposed and standard wall functions were applied (Launder and Spalding, 1974).

The simpleFoam solver was used. Second order numerical schemes were chosen for the discretization due to their accuracy and stability. For the convective terms, a linear upwing scheme was used, whereas the linear corrected scheme
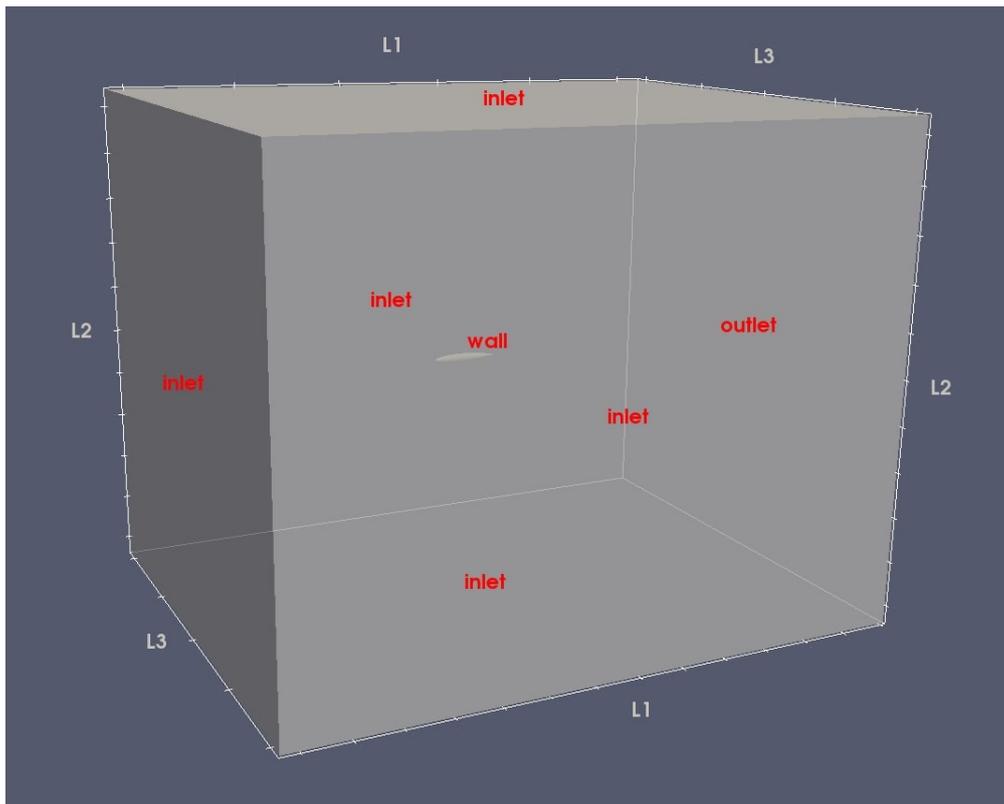
Figure 2: Computational domain and boundary conditions.

was used for the diffusive terms and a linear scheme was set for the gradient terms. The simulations were run using the GAMG multigrid solver for the pressure and the Gauss Seidel smooth solver for the velocity and for the turbulence parameters. The convergence of the simulation was verified by assuring a residual drop of 10^{-6} and a variation of the resistance of less then 0.1% between subsequent iterations.

3. Results and Discussion

3.1 Sensitivity to size of the domain

Sensitivity tests for the size of the domain were also addressed. In this sense, three domains were considered with dimensions described in Table 1 (see also Fig. 2), divided by the length of the hull model. The calculated drag coefficients are shown in the same table. We can see that the differences between medium and large domains are about 0.65%. As consequence, we have considered the medium size domain as the best compromise option between accuracy and computational cost. This domain size is used for the following analyses.

Table 1: Dimensions of the studied computational domains.

	L_1/L	L_2/L	L_3/L	$c_t \times 10^3$
Small	5	3	3	3.399
Medium	10	8	8	3.201
Large	15	13	13	3.180

3.2 Sensitivity to grid refinement

In order to assess the influence of grid size on the drag coefficient, three grids were analysed. A representative grid size, according to Celik *et al.* (2008), was defined for each mesh and the grid refinement factor was kept higher than 1.3 between grids. The number of volumes, maximum and average y+ values and resulting c_t values are summarized in Tab. 2 for two Reynolds numbers. Results suggest an oscillatory grid convergence, and differences in c_t between medium and fine meshes are smaller than 2%. Results from the medium meshes were considered acceptable and these grids were used for the succeeding analyses.
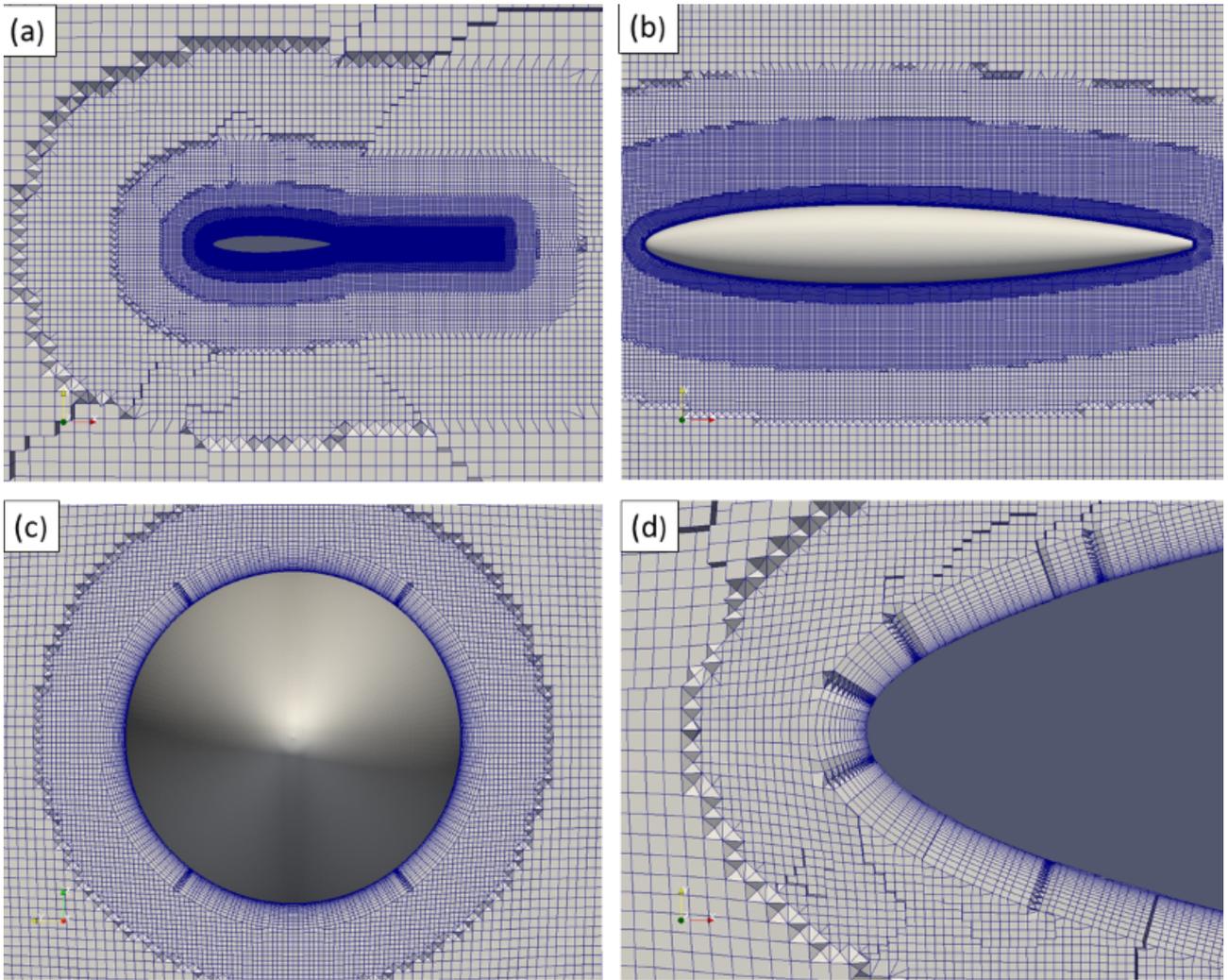
Figure 3: Imagens of the used grid. (a) xy view, (b) mesh around surface, (c) yz plane view near surface, (d) detail of layers close to the leading edge.

Table 2: Mesh information and results of the grid refinement analysis.

Reynolds	Mesh	Number of Volumes	$c_t \times 10^3$	Maximum y^+	Average y^+
	Coarse	3156471	3.440	124.39	91.72
4.0×10^6	Medium	7365473	3.201	103.20	72.90
	Fine	11676278	3.266	86.55	61.06
	Coarse	3282863	2.959	124.63	96.81
2.5×10^7	Medium	8215133	2.781	96.80	72.27
	Fine	12897238	2.825	80.14	60.64

3.3 Sensitivity to turbulence boundary conditions

Assessing the influence of the turbulent properties on the boundary conditions is also important, as they can influence the results (Spalart and Rumsey, 2007). As the experimental tests from Gertler (1950) consisted of towing the model in a test tank, we considered the freestream condition in the numerical model should present low turbulence levels. In this sense, based on the literature (Kouh *et al.*, 2009; Spalart and Rumsey, 2007; Mahon and Zhang, 2005), baseline values of $I = 0.5\%$ and $\nu_t/\nu = 10$ were chosen to compute k and ω values at the inlet boundaries. Results of sensitivity tests are shown in Tab. 3. Convergence could not be achieved for $I = 0.1\%$, however the total drag coefficient varied less than 1% with turbulence intensity and viscosity ratio in the tested range.

Table 3: Results of the analysis of the influence of turbulence parameters.

	ν_t/ν	k	ω	$c_t \times 10^3$
$I = 0.5\%$	$\nu_t/\nu = 5$	0.000084375	16.875	3.201
	$\nu_t/\nu = 10$	0.000084375	8.4375	3.235
	$\nu_t/\nu = 15$	0.000084375	5.625	3.235
$I = 1\%$	$\nu_t/\nu = 5$	0.0003375	67.50	3,230
	$\nu_t/\nu = 10$	0.0003375	33.75	3.234
	$\nu_t/\nu = 15$	0.0003375	22.5	3.235

3.4 Validation and discussion

Numerical results of the total drag coefficient (c_t) are depicted in Fig. 4 and compared to experimental data. In this plot, the original experimental results from Gertler (1950) were corrected by subtracting the influence of the sand strip and towing struts. Results show reasonable agreement with experimental data. The total drag coefficients were underpredicted for $Re < 1.25 \times 10^7$ and overpredicted for $Re > 1.5 \times 10^7$. Maximum deviations were about 8.8% for $Re = 4 \times 10^6$ and 7.2%. It is important to emphasize, however, that the c_t values for Re close to 1×10^7 are strongly influenced by the wave drag component (Gertler, 1950), which has not been taken into account in the simulations.
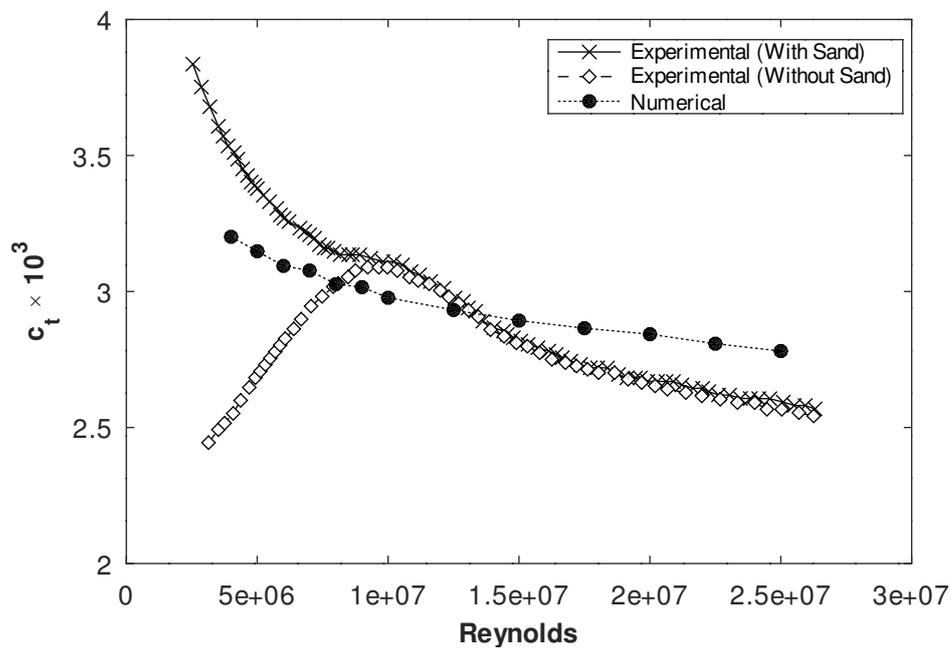

Figure 4: Drag coefficients per Reynolds number. Comparison between numerical results and corrected experimental data.

In order to identify possible sources of error, the contributions of the friction drag and pressure drag on the total drag coefficient are separated and shown in Fig. 5a. It is clear that the friction drag is the main contributor to total drag. Figure 5b depicts the comparison between the numerical friction coefficient (c_f) and the friction coefficient calculated by the Schoenherr's expression (Gertler, 1950) and ITTC-1957 (Morrall, 1970). We can see that the predicted c_f deviates from the calculated by the well-known analytical expressions.

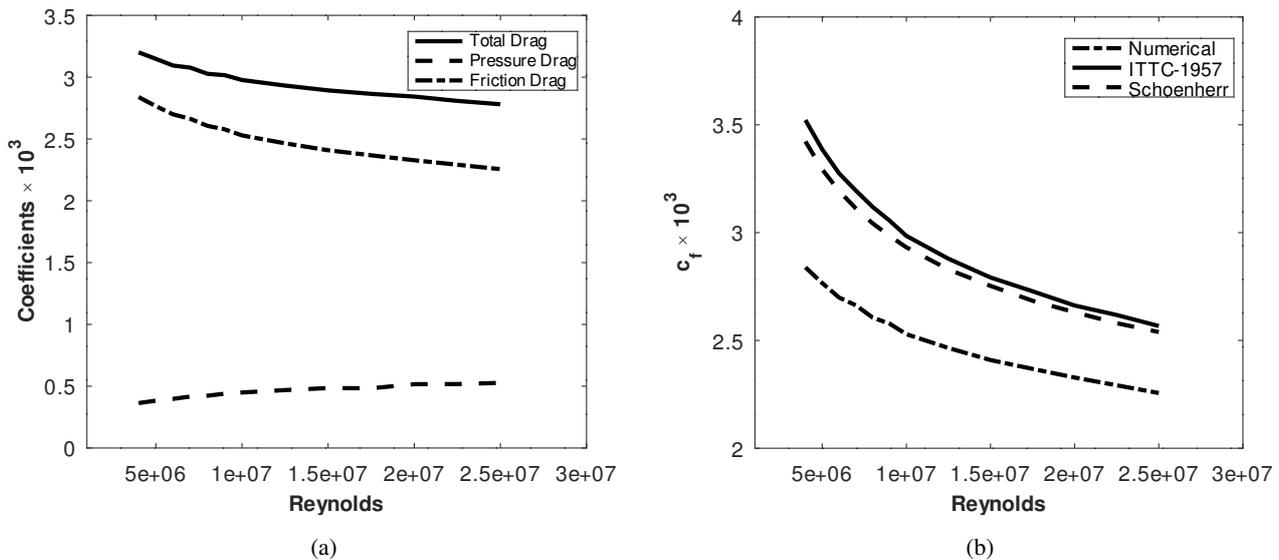(a)   (b)

Figure 5: Drag coefficients per Reynolds number. (a) Separated contributions of friction and pressure drag coefficients compared to the total drag coefficient. (b) Numerical friction coefficient compared to those obtained by the Schoenherr formula (Gertler, 1950) and the ITTC-1957 (Morrall, 1970) method.

4. CONCLUSIONS

A simulation model was built using the code OpenFOAM in order to predict drag on submarines. Sensitivity tests for domain size, grid refinement and boundary conditions were conducted. For this analysis, a simple axisymmetric geometry was chosen for validation.

Results of total drag coefficients were in reasonable agreement with experimental data. Drag was overprectited for the higher Reynolds range with maximum deviations of about 8.8%. Comparisons of the CFD-computed friction coefficients with well-known analytical formulas indicated possible uncertainties in the prediction of this coefficient. Nevertheless, the differences between the simulation, with fixed model immersed in freestream, and the experiments, with the submarine towed in the test tank, could also be a cause of the observed deviations. The presence of tank walls, free surface and towing struts are not reproduced by the simulation model. Although the correction for the influence of the towing struts presented by Gertler (1950) was used, it is still a possible source of uncertainties.

More effort is required to improve the simulation model. However, despite of the observed deviations, the used CFD software and mesh generation tool provide a suitable framework for analyzing complex geometries and coupling with optimization routines, being a potential tool for the conceptual/preliminary design of the propulsion system of submarines.

5. ACKNOWLEDGEMENTS

6. REFERENCES

Celik, I.B., Ghia, U., Freitas, C.J., Coleman, H. and Raad, P.E., 2008. "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications". *Journal of Fluids Engineering*, Vol. 130, No. 7. ISSN 0098-2202. doi:10.1115/1.2960953.

Fureby, C., Anderson, B., Clarke, D., Erm, L., Henbest, S., Giacobello, M., Jones, D., Nguyen, M., Johansson, M., Jones, M., Kumar, C., Lee, S.K., Manovski, P., Norrison, D., Petterson, K., Seil, G., Woodyatt, B. and Zhu, S., 2016. "Experimental and numerical study of a generic conventional submarine at 10° yaw". *Ocean Engineering*, Vol. 116, pp. 1 – 20. ISSN 0029-8018. doi:10.1016/j.oceaneng.2016.01.001.

Gertler, M., 1950. "Resistance Experiments on a Systematic Series of Streamlined Bodies of Revolution - For Application to the Design of High-speed Submarines". Technical report, Navy Department, David W. Taylor Model Basin.

Holloway, A., Jeans, T. and Watt, G., 2015. "Flow separation from submarine shaped bodies of revolution in steady turning". *Ocean Engineering*, Vol. 108, pp. 426–438. ISSN 00298018. doi:10.1016/j.oceaneng.2015.07.052.

Jackson, H.A., 1992. "Fundamentals of Submarine Concept Design". *Trans. Soc. Of Naval Archit. & Marine Engr*, Vol. 100, pp. 419–448.

Joubert, P.N., 2004. "Some Aspects of Submarine Design Part 1. Hydrodynamics". Technical report.

Joubert, P.N., 2006. "Some Aspects of Submarine Design Part 2. Shape of a Submarine 2026". Technical report.

Kouh, J.S., Chen, Y.J. and Chau, S.W., 2009. "Numerical study on scale effect of form factor". *Ocean Engineering*, Vol. 36, No. 5, pp. 403–413. ISSN 00298018. doi:10.1016/j.oceaneng.2009.01.011.

Landweber, L. and Gertler, M., 1950. "Mathematical Formulation of Bodies of Revolution". Technical report.

Larsen, C.A., 1955. "Additional Tests of Series 58 Forms. Part I. Resistance Tests of a Parallel Middle Body Series:". Technical report, Defense Technical Information Center, Fort Belvoir, VA. doi:10.21236/ADA951810.

Launder, B.E. and Spalding, D.B., 1974. "The numerical computation of turbulent flows". *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, No. 2, pp. 269 – 289. ISSN 0045-7825. doi:https://doi.org/10.1016/0045-7825(74)90029-2.

Liu, H. and Huang, T.T., 1998. "Summary of DARPA Suboff Experimental Program Data." Technical Report Report No.: CRDKNSWC/HD-1298-11., Naval Surface Warfare Center, Maryland.

Mahon, S. and Zhang, X., 2005. "Computational Analysis of Pressure and Wake Characteristics of an Aerofoil in Ground Effect". *Journal of Fluids Engineering*, Vol. 127, No. 2, pp. 290–298. ISSN 0098-2202, 1528-901X. doi: 10.1115/1.1891152.

Menter, F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA Journal*, Vol. 32, No. 8, pp. 1598–1605. doi:10.2514/3.12149.

Morrall, A., 1970. "1957 ITTC model ship correlation line values of frictional resistance coefficient". Technical Report NPL Ship Report 142, National Physical Laboratory, Ship Division.

Mulvihill, L.P. and Yang, C.I., 2007. "Numerical Simulation of Flow over Fully Appended ONR Body-1 with Overset Grid Scheme". Ann Arbor, Michigan, p. 13.

Pan, Y.c., Zhang, H.x. and Zhou, Q.d., 2012. "Numerical Prediction of Submarine Hydrodynamic Coefficients using CFD Simulation". *Journal of Hydrodynamics*, Vol. 24, No. 6, pp. 840–847. ISSN 1001-6058, 1878-0342. doi: 10.1016/S1001-6058(11)60311-9.

Randwijck, E.F.B. and Feldma, J.P., 2000. "Results of experiments with a segmented model to investigate the distribution of the hydrodynamic forces and moments on a streamlined body of revolution at an angle of attack or with a pitching angular velocity." Technical Report Report No.: NSWCCD-50-TR-2000/008, Naval Surface Warfare Center, Maryland.

Spalart, P.R. and Rumsey, C.L., 2007. "Effective Inflow Conditions for Turbulence Models in Aerodynamic Calculations". *AIAA Journal*, Vol. 45, No. 10, pp. 2544–2553. doi:10.2514/1.29373.

The OpenFOAM Foundation, 2018. "OpenFOAM v6 User Guide". URL https://cfd.direct/openfoam/user-guide-v6.

Zhang, J., Maxwell, J.A., Gerber, A.G., Holloway, A.G.L. and Watt, G.D., 2013. "Simulation of the flow over axisymmetric submarine hulls in steady turning". *Ocean Engineering*, Vol. 57, pp. 180–196. ISSN 00298018. doi: 10.1016/j.oceaneng.2012.09.016.

7. RESPONSIBILITY NOTICE

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