26th International Congress of Mechanical Engineering
COBEM 2021
ABCM Associação Brasileira de Engenharia e Ciências Mecânicas
November 22-26, 2021, Virtual Congress, Brazil

# COB-2021-0409
# COMPARISON BETWEEN AXISYMMETRIC AND 3D MODELS FOR THE EVALUATION OF DRAG ON SUBMARINE HULLS

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

***Abstract.*** *This paper presents a comparison between results obtained with a 3D and an axisymmetric model of a submarine hull. The objective is to obtain an accurate model with low computational cost for such vessels. The chosen geometry is the hull model 4165 from series 58. Numerical errors due to grid refinement and sensitivity to boundary conditions are assessed and validation through comparisons with experimental data is provided. The analyzed Reynolds numbers range from* $4\times10^6$ *to* $2.5\times10^7$*. Results obtained with the axisymmetric grid showed deviations of about 3%, when compared to the experimental data. In contrast, for a 3D model presented in a previous publication, such deviations were up to 8.8%. Even though axisymmetric models may not be useful for analyzing more complex geometries, their lower computational cost and simplified mesh generation process make them suitable for optimization routines. Such simulations can also provide preliminary estimates for the design of the propulsive and maneuvering systems of submarines.*

***Keywords:*** *Drag computation, Hull design, Submarine, OpenFOAM*

## 1. INTRODUCTION

Due to their restricted application and the confidential nature of these projects, studies describing models to predict drag of submarines are extremely rare in the literature. Many studies in the literature have proposed hull geometries for submarines aiming at improving the internal design and reducing drag.

Landweber and Gertler (1950) made important contributions to submarine design proposing the teardrop-shaped hull, Jackson (1992) proposed an alternative formulation to teardrop-shaped hull, Joubert (2004, 2006) shedded some light on submarine hydrodynamics, proposing a hull with elliptical nose and parabolic tail. Larsen (1955), Randwijck and Feldma (2000) and Fureby *et al.* (2016) reported experimental data of drag at deep submergence for bodies of revolution, 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). Liu and Huang (1998) 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.

Mulvihill and Yang (2007) applied a Reynolds-averaged Navier-Stokes (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 model, Zhang *et al.* (2013) and Holloway *et al.* (2015) performed RANS models to obtain the force and momentum coefficients for DRDC STR, DARPA Suboff and DTRC 4621 submarines, Kouh *et al*. (2009) used a RANS model to investigate the effect of scaling on the shape factor of a DTRC 4621 submarine, Fureby *et al.* (2016) investigated the flow over a DSTO submarine with appendices under different yaw angles.

Furthermore, it is well-known that the development of simulation models is a difficult and methodical task, which requires an extensive process of validation and verification. During the last decades, numerical methods have been used for the performance analysis of complex engineering systems and its use has become more usual during the preliminary design stages of any project. Aiming to reduce computational cost in these stages, several problems can be simplified and modeled by two-dimensional simulations, which usually require less computational resources compared to their three-dimensional counterparts. Indeed, 2D simulations can provide reasonable approximations in many situations, such as simulations of flow around cylinders, axisymmetric bodies or airfoils (Sharma *et al.,* 2019).

In the literature, comparisons between 3D and 2D simulations can be found in the most diverse areas. Salehi and Rahimi (2008) reported a comparison between 2D axisymmetric and 3D transient flow simulation of bubble column reactors. Pérez and Vakkilainen (2019) conducted a comparison between turbulence models in two and three dimensional meshes for ash deposition models. Pashchenko (2017) compared hydrogen/air combustion CFD-modeling for 3D and 2D computational domains. Misha *et al.* (2014) performed simulations for the comparison of tray dryer systems using 2D and 3D models. Li *et al.* (2014) studied CFD simulations of circulating fluidized bed risers and evaluated the differences between 2D and 3D simulations.

The objective of this work is to validate an axisymmetric CFD model, using the open source code OpenFOAM (The OpenFOAM Foundation, 2018), for the prediction of drag of submarines and compare its accuracy with the 3D model from da Silva *et al.* (2020). In section 2, information of the reference geometry, experimental data, 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

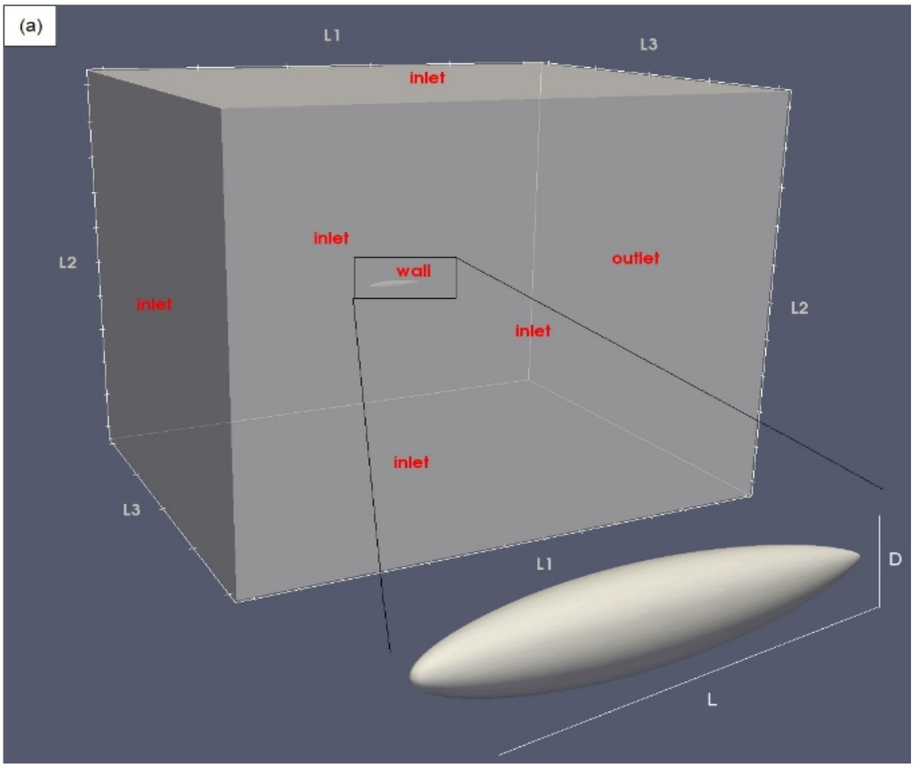
The same geometry chosen by da Silva *et al*. (2020) was used: the 4165 model of series 58 by Gertler (1950). Its main geometric characteristics are: length L = 2.74m, maximum diameter D = 0.392m and wetted area S = 2.491m². Figure 1a depicts the 3D model (da Silva et al., 2020) and 1b shows the slice used for the axisymmetric simulations.

In the series 58 experiments performed by Gertler (1950), a parametric analysis of the dimensions of submarine hulls was carried out. Each hull was towed at different speeds and depth levels. Furthermore, some of the experiments used sand strips close to the leading edge of the models to guarantee a turbulent boundary layer. The influence coefficients of the sand strip and also of the towing structures are evidenced in the work of Gertler (1950), and were used to correct the experimental data before the comparisons with the numerical data.

In the present study, results from a 2D model are compared with the results from da Silva *et al*. (2020), for the 3D model, and with the corrected experimental data from Gertler (1950).

### 2.2 Simulation Model

Similar to the 3D case, for the 2D axisymmetric, the simulations were conducted using incompressible Reynolds Averaged Navier-Stokes (RANS) with the SST turbulence model (Menter, 1994) implemented in OpenFOAM v8.0. The computational domain and boundary conditions for both the 3D and axisymmetric models are depicted in Fig. 1.



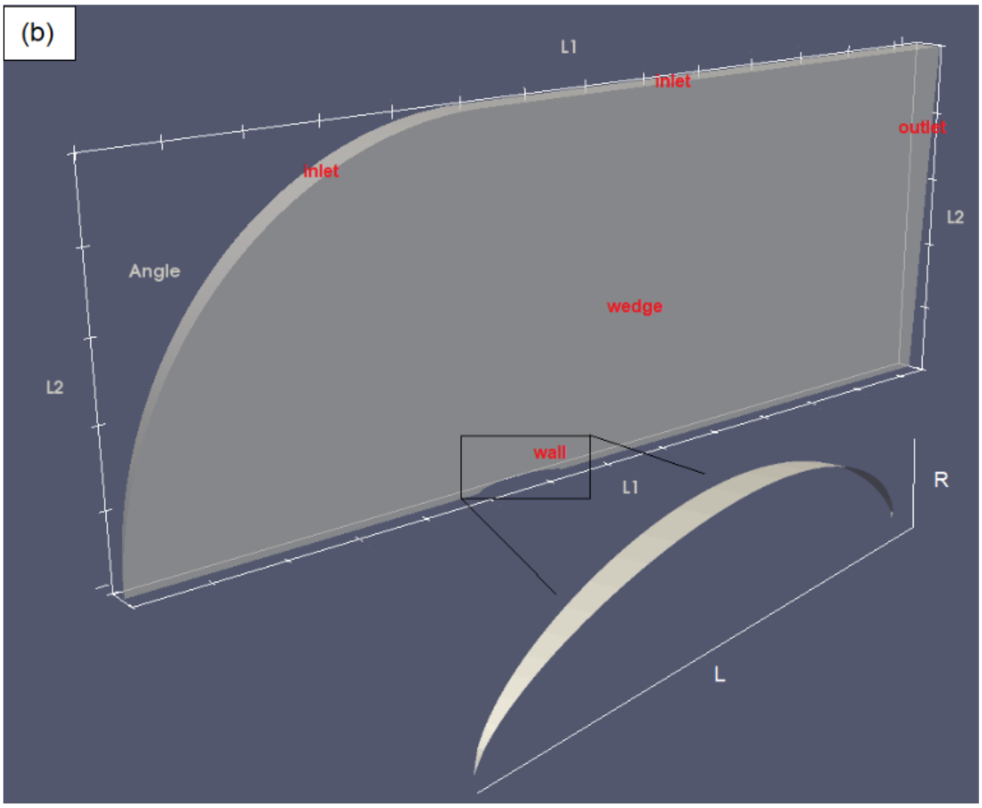



Figure 1: Computational domain, boundary conditions and modeled geometry of hull 4165 from series 58 Gertler (1950). (a) 3D Model, (b) Axisymmetric Model.

Differently from the domain analyzed in the 3D case, in the frontal region of the axisymmetric domain, a circumference was used in order to better adapt the cells to the curvature of the leading edge of the profile. In addition, for the front and back faces, the wedge condition was used, as required for axisymmetric simulations in OpenFOAM (The OpenFOAM Foundation, 2018).

The size of the domain corresponds to the medium size from the 3D case (L1/L = 10 and L2/L = 8). According to the OpenFOAM documentation, for the use of the wedge condition, the angle of wedge must be less than or equal to 5°. In order to assess the sensitivity to these parameters, simulations were performed with three different wedge angles (Figure 3) and results are presented in section 3. The mesh constructed in Gmsh (Geuzaine and Remacle, 2009) is formed by structured hexahedral volumes. As the proposal is to use a 2D approach, only one layer of elements are generated in the azimuthal direction. On the other hand, the 3D model from da Silva *et al.* (2020) used a mesh built predominantly of cubic elements, with an octree refinement structure. Layers of prismatic volumes are added near walls for the discretization of the boundary layer gradients. A comparison of both grids is shown in Figure 2.

A freestream boundary condition was imposed in the frontal and upper side 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 = 0.1% and eddy viscosity ratio νt/ν = 5. Sensitivity tests for the turbulence variables are presented in section 3. For the outflow condition, a prescribed back pressure value of 0 Pa was imposed and a zero gradient condition was used for the other properties.

On the surface of the model, the no-slip boundary condition was imposed. For the sensitivity analysis, wall functions were used (Launder and Spalding, 1974). For the final simulations, to obtain the total drag coefficient ($c_t$), the boundary layer was resolved up to the wall, and a value of y+ less than 1 was ensured for the near-wall cells.

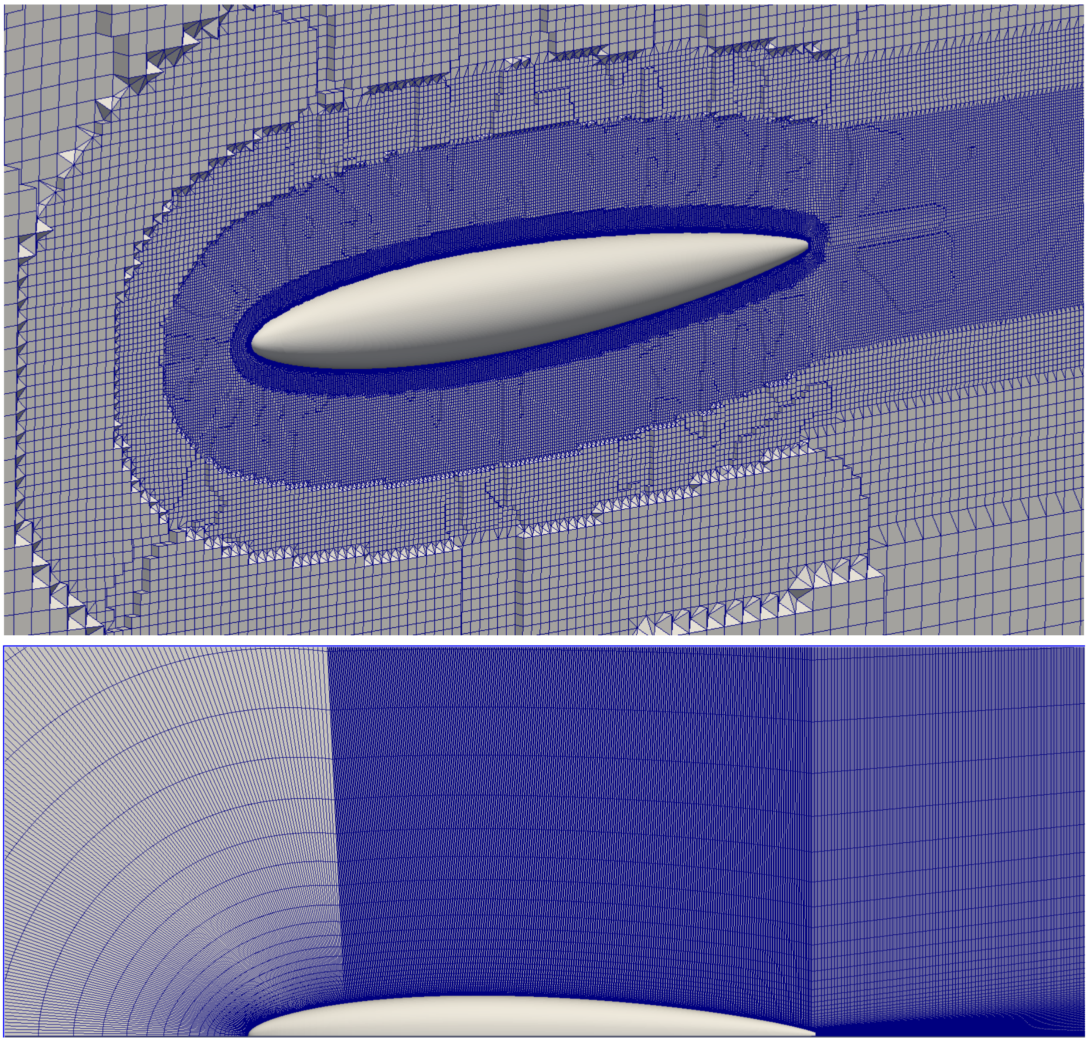


Figure 2: 3D (top) and Axisymmetric (bottom) grid comparison.

The simpleFoam solver was used with the aid of the mapFields function to initialize the fields and interpolate results between meshes. The numerical schemes for the axisymmetric model are the same as the 3D model, i.e. second order numerical schemes for the discretization. For the convective terms, the linear upwind scheme was used, whereas the linear corrected scheme was used for the diffusive terms and the 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. For the solution, the convergence of the axisymmetric simulation was verified by asserting a residual drop of at least $10^{-5}$ for the SIMPLE iterations and a variation on the drag coefficient of less than 0.01% between subsequent iterations.

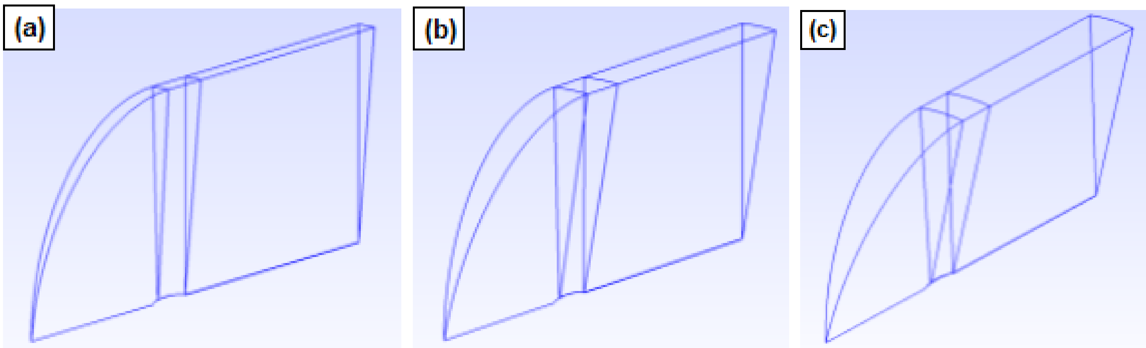



Figure 3: Axisymmetric meshes according to wedge angle. (a) 5°, (b) 10°, (c) 15°.

With regard to mesh blocking, four main regions were defined: profile leading edge, middle body, parallel trailing edge and perpendicular trailing edge. In Figure 4a these regions can be observed, also, the block depicted in Figure 4a was the block used for the sensitivity analysis of grid refinement and turbulent parameters.

Subsequently, to obtain the total drag coefficients, another block structure was used (Figure 4b). This structure provides a better control of the discretization in the near-wall region, in order to guarantee y+ less than one for the first layer of volumes.

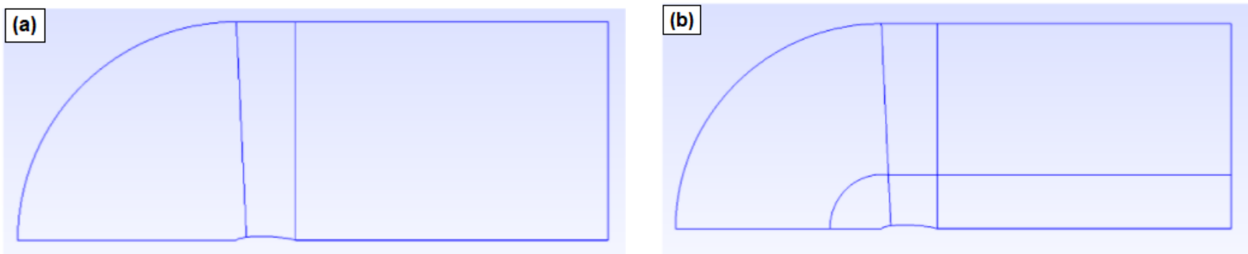



Figure 4: Proposed grid blocks for the axisymmetric case. (a) First used block structure, (b) Second used block structure.

## 3. Results and Discussion

### 3.1 Sensitivity to grid refinement and wedge angle

Sensitivity tests for wedge angle and mesh refinement were conducted. Three wedge angles were considered (Figure 3) and three levels of grid refinement for each angle. 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 results of the total drag coefficient, number of volumes, maximum and average y+ for all cases are in tables 1, 2 and 3. Only the highest Reynolds number was analyzed. Note that the differences in the $c_t$ results between each wedge angle were below 0.45%. For the refinement levels, the differences were below 14% between the medium and the coarse mesh, below 1.7% between the medium mesh and fine mesh, and 0.9% between the fine and very fine mesh. Based on these results, the use of a wedge angle of 5° and the very fine grid were considered for the subsequent analyses.

Table 1: Grid refinement analysis for the wedge angle of 5°.

| **Reynolds** | **Mesh** | **Number of Volumes** | $c_t \times 10^3$ | **Maximum y+** | **Average y+** |
|---|---|---|---|---|---|
| $2.5 \times 10^7$ | Coarse | 129873 | 2.258 | 283.54 | 217.84 |
| | Medium | 233694 | 2.599 | 168.12 | 135.27 |
| | Fine | 308017 | 2.642 | 102.09 | 55.38 |
| | Very Fine | 655559 | 2.666 | 0.751 | 0.590 |

Table 2: Grid refinement analysis for the wedge angle of 10°.

| **Reynolds** | **Mesh** | **Number of Volumes** | $c_t \times 10^3$ | **Maximum y+** | **Average y+** |
|---|---|---|---|---|---|
| $2.5 \times 10^7$ | Coarse | 129873 | 2.254 | 280.85 | 216.37 |
| | Medium | 233694 | 2.593 | 167.50 | 134.89 |
| | Fine | 308017 | 2.636 | 102.29 | 55.24 |

Table 3: Grid refinement analysis for the wedge angle of 15°.

| **Reynolds** | **Mesh** | **Number of Volumes** | $c_t \times 10^3$ | **Maximum y+** | **Average y+** |
|---|---|---|---|---|---|
| $2.5 \times 10^7$ | Coarse | 129873 | 2.250 | 276.36 | 213.90 |
| | Medium | 233694 | 2.588 | 166.81 | 134.35 |
| | Fine | 308017 | 2.632 | 102.62 | 55.00 |

### 3.2 Sensitivity to turbulence boundary conditions

Similar to the 3D case, the sensitivity to turbulence inflow conditions was analysed for the axisymmetric case. Assessing the influence of the turbulent properties is important, as they can influence the results (Spalart and Rumsey, 2007). Based on the literature (Kouh et al., 2009; Spalart and Rumsey, 2007; Mahon and Zhang, 2005), a range of values of I = 0.1%, 0.5%, 1% and νt/ν = 5, 10, 15 were chosen to compute k and ω values at the inlet boundaries. Results of sensitivity tests are shown in Tab. 4. Turbulent intensities values of 0.5% and 1% resulted in divergence, while the 0.1% tests resulted in much more stable simulations with low sensitivity to the viscosity ratio. As a result, the turbulent intensity of 0.1% and eddy viscosity ratio of 5 was chosen for the next axisymmetric simulations.

Table 4: Grid refinement analysis for the wedge angle of 15°.

| I | νt/ν | k | ω | $c_t \times 10^3$ |
|---|---|---|---|---|
| I = 0.1% | νt/ν = 5 | 0.0001286214 | 25.72428 | 2.599 |
| | νt/ν = 10 | 0.0001286214 | 12.86214 | 2,599 |
| | νt/ν = 15 | 0.0001286214 | 8.57476 | 2.599 |
| I = 0.5% | νt/ν = 5 | 0.003215535 | 643.107 | Diverged |
| | νt/ν = 10 | 0.003215535 | 321.5535 | 2.598 |
| | νt/ν = 15 | 0.003215535 | 214.369 | 2.599 |

### 3.3 Validation and discussion

Numerical results of the total drag coefficient of both 3D and axisymmetric cases are compared to the experimental data in Figure 5. In this plot, the experimental results from Gertler (1950) were corrected by subtracting the influence of the sand strip and towing struts.

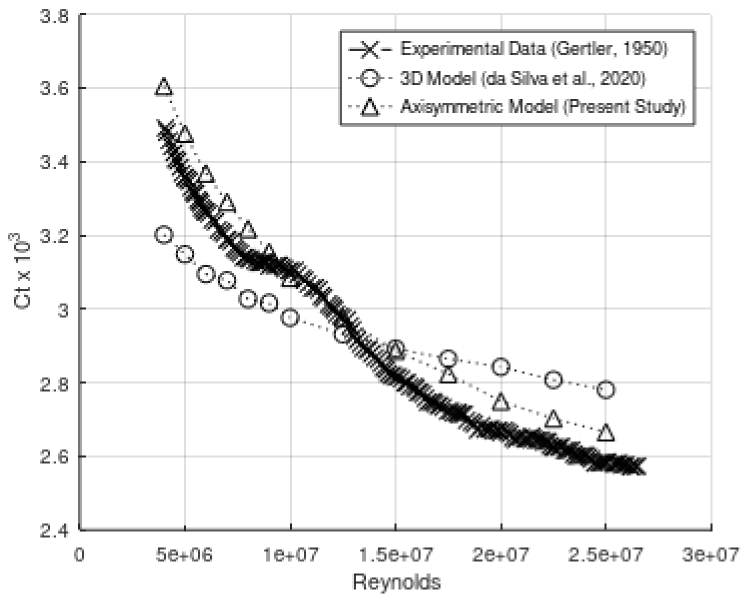



Figure 5: Total drag coefficients per Reynolds number: numerical results and corrected experimental data.

Note that results from axisymmetric model shows a better prediction of the trends of $c_t$ with Reynolds number, keeping the deviations from the experimental values in about 3% along the curve. Disparities from the experimental data are even lower in the region from 8e+6 to 1.5e+7. However, in the experiments, wave drag is significant in this range of velocities and has not been taken into account in the numerical model. On the other hand, for the 3D case, the most significant deviations vary from 7% to 8%.

The friction and pressure drag components were separated for both models (Figure 6a and Figure 6b). The friction drag calculated according to ITTC-1957 (Morrall, 1970) was also added for comparison. Note that the friction drag coefficient of the axisymmetric case is much closer to the ITTC-1957 method than the 3D case. The 3D case underestimated the friction drag, which is the main contributor of the total drag. For the pressure drag coefficient, the results between cases differ again in both level and trends with Reynolds number. In fact the mesh generation process of the 2D case allowed a greater control of the mesh in the near wall region, making it easier to reach y+ < 1 for volumes adjacent to the wall, in comparison to the prism layer generation process of the snappyHexMesh tool, used in the 3D approach. In this sense, we suppose the differences in the near-wall mesh and modeling could be the main cause of the better performance of the 2D-axisymmetric model.

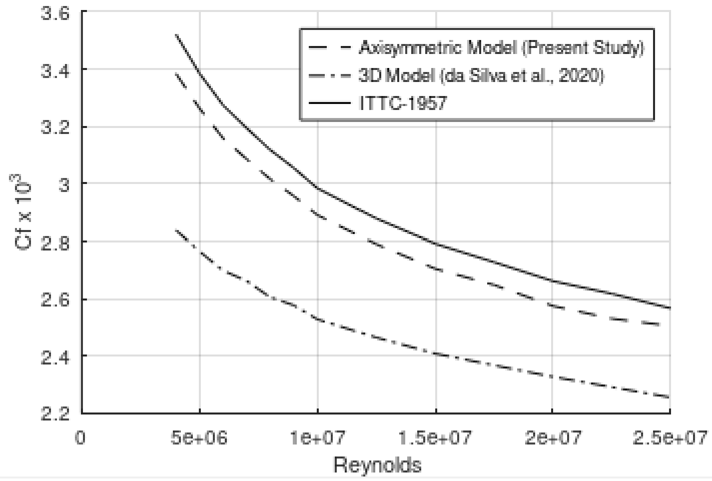



Figure 6a: Drag coefficients per Reynolds number. Separated contributions of friction drag coefficients compared to the ITTC-1957 (Morrall, 1970) method.

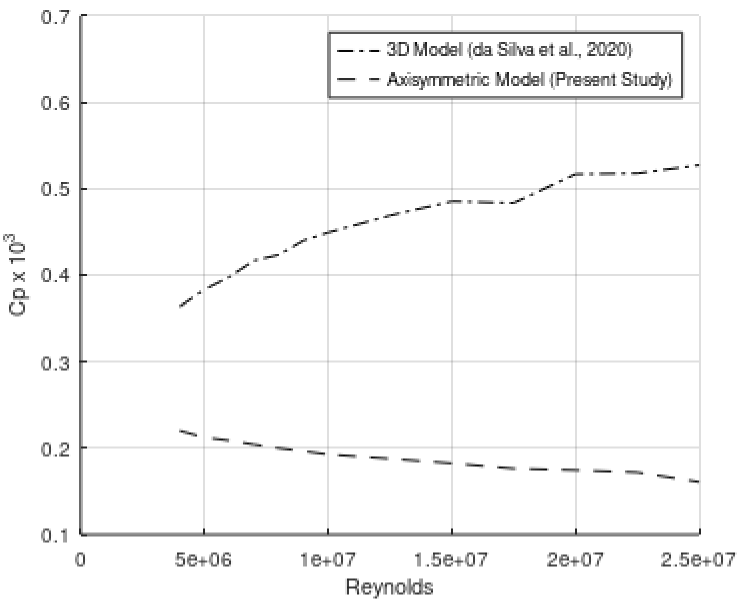



Figure 6b: Drag coefficients per Reynolds number. Separated contributions of pressure drag coefficients.

Another interesting parameter for comparison is the simulation running time. Both 3D and axisymmetric cases were run using 16 cores and the total simulation time (in hours) is shown in Figure 7 for each Reynolds number. It is clear, as expected, that the axisymmetric model required less simulation time to achieve converged results. The 2D model is relatively more refined than the 3D model, since the simulations were conducted for y+ < 1, if the same had been done for the 3D mesh, the time differences would be even greater.

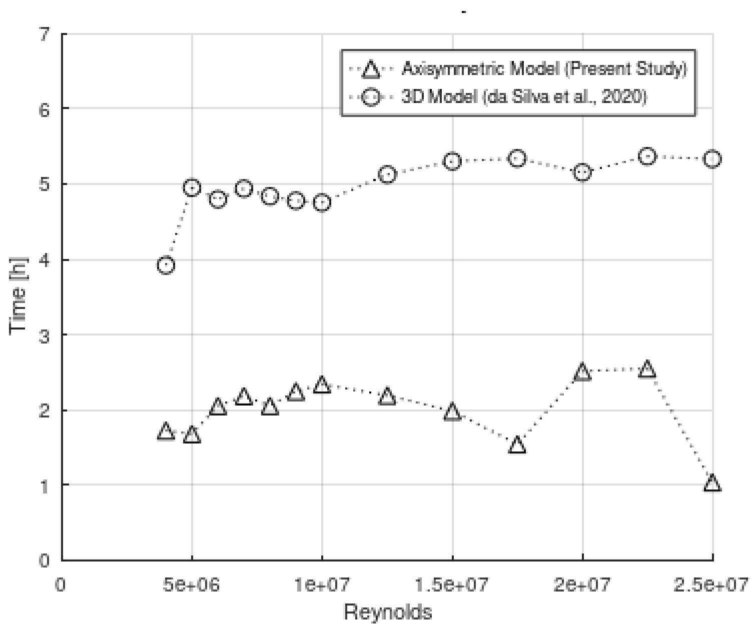



Figure 7: Total time of execution (in hours) per Reynolds number.

## 4. CONCLUSION

An axisymmetric simulation model was built using the OpenFOAM code in order to improve the prediction of drag on submarine hulls, in comparison to a previously used. Sensitivity tests for grid refinement, wedge angle and boundary conditions were conducted for the axisymmetric case. For the 3D case paper da Silva et al. (2020) concluded that the friction drag coefficient was not being adequately predicted. The results of the present study confirmed that these suspicions were correct, since the axisymmetric model predicts this coefficient with more certainty as it is closer to the ITTC-1957 method. In addition, the pressure-drag coefficients of both cases differ. The axisymmetric case proved to be satisfactory for the analyses, in addition to being fast from the point of view of computational cost, being a good tool for preliminary analyzes and highly automatable in optimization routines. For the 3D case, it was concluded that it should be revisited for possible improvements in the prediction of both pressure and friction drag. These improvements can be in the mesh quality, trying to achieve the same level of refinement in the boundary layer of the 2D case. Nevertheless, the differences between the simulation, with a 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. Finally, the used CFD model stands as a potential tool for the conceptual/preliminary design of the propulsion system of submarines.While the 2D model has lower computational cost and is suitable for optimization of simplified geometries, the 3D model is capable of handling more complex geometries with inclinations and appendices.

## 5. 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.

da Silva, F. D., Ribeiro, E. B., Tancredi, T. P. and Silva, E., 2020. “Prediction of Drag on Submarines Using OpenFOAM”. Conference: ENCIT 2020 - 18th Brazilian Congress of Thermal Sciences and Engineering. doi: 10.26678/ABCM.ENCIT2020.CIT20-0513.

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.

Geuzaine, C. and Remacle, J.-F., 2009. “Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities”. International Journal for Numerical Methods in Engineering 79(11), pp. 1309-1331.

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.
Li, T., Pannala, S. and Shahnam, M., 2014. "CFD simulations of circulating fluidized bed risers, part II, evaluation of differences between 2D and 3D simulations." Powder Technology, Vol. 254, pp. 115-124. ISSN 0032-5910. doi: 10.1016/j.powtec.2014.01.022.
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.
Misha, S., Mat, S., Ruslan, M. H., Sopian, K. and Salleh, E., 2014. "Comparison Between 2D and 3D Simulations of a Tray Dryer System Using CFD Software". World Applied Sciences Journal, Vol. 29, Issue 10, pp. 1301-1309. ISSN 1818-4952. doi: 10.5829/idosi.wasj.2014.29.10.1581.
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.
Pashchenko, D., 2017. "Comparative analysis of hydrogen/air combustion CFD-modeling for 3D and 2D computational domain of micro-cylindrical combustor." International Journal of Hydrogen Energy, Vol. 42, Issue 49, pp. 29545-29556. ISSN 0360-3199. doi: 10.1016/j.ijhydene.2017.10.070.
Pérez, M. G. and Vakkilainen, E., 2019. "A comparison of turbulence models and two and three dimensional meshes for unsteady CFD ash deposition tools." Fuel, Vol. 237, pp. 806-811. ISSN 0016-2361. doi: 10.1016/j.fuel.2018.10.066.
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.
Salehi, M. A. and Rahimi, R. 2008. "Comparison Between 2D and 3D Transient Flow Simulations of Gas – Liquid Dynamics in Two Phase Cylindrical Bubble Column Reactors by CFD". International Journal of Chemical Science, Vol. 6, ISSN 857-872.
Sharma, G., Gupta, P.C. and Kant, S., 2019. "CFD Analysis of Two Dimensional Subsonic Flow Over S1223RTL Airfoil". International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Vol. 8, Issue-3S.
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.

## 6. RESPONSIBILITY NOTICE

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