DES-based computation of the flow around the DARPA suboff

A Lungu
"Dunarea de Jos" University of Galati, Faculty of Naval Architecture, Department of Naval Architecture, Domneasca Street No. 47, Galati 800008, Romania
E-mail: adrian.lungu@ugal.ro

Abstract. Flow modelling is a key issue not only for the correct prediction of hull-propeller interactions, manoeuvring characteristics and the flow field in the stern region of any marine vehicle, but also for the correct estimation of the fuel consumption. The paper describes a thorough investigation of the hydrodynamic performances of the DARPA suboff hull. The flow is numerically simulated by integrating in time the unsteady Navier-Stokes equations in which closure to the turbulence is achieved by means of a modified detached eddy simulation (DES hereafter) technique. The solver used for the purpose is the ISIS-CFD, part of the Numeca Fine™/Marine suite, which employs the finite volume of fluid technique. A comparative analysis between the numerical solutions based on the DES and explicit algebraic stress model (EASM hereafter) are provided in an attempt to sustain the choice for the turbulence model in the present study. Extensive parallel computations are carried out on 120 processors for which comparisons with the experimental data prove the accuracy of the chosen investigation methodology. A grid convergence test is performed for verification and validation purposes.

1. Introduction

Appropriate hydrodynamic design of an underwater vehicle is probably the most important issue for achieving a suitable and effective performance. The proper design of the external geometric configuration of the vehicle is also essential for minimizing the energy consumption required to power the vehicle for the desired range of operation and at the requested speed. An improper body geometry can easily lead to unwanted high drag, noise, and instability. Stability and control of the underwater vehicles are issues that generally require close attention. Obviously, to determine accurately the stability, control and manoeuvring characteristics of a submerged vehicle, it is required to properly predict the imposed hydrodynamic forces and moments under various operating conditions.

The modern underwater vehicles have unconventional appendages to achieve high manoeuvrability performances at intermediate to high Reynolds numbers [1, 2]. This raises two challenges for a full-scale simulation of the flows around the underwater vehicles: the first one is to handle the complex geometric and moving boundaries; the second one is to calculate the characteristics of viscous flows near the boundaries and in the wake [3, 4]. For this purpose, computational fluid dynamics (CFD hereafter) methods have been widely used for simulating the flow around the submarines including the interactions between the hull and the propeller. The complexity of the work being done, the variability and the difficulty of environmental conditions is gaining importance in the design phase of the vehicle [5]. Hydrodynamic design of submarine propellers has also been an important issue for the designers. A submarine propeller has to be designed and optimized by considering the propeller-hull interaction. Finite volume method based commercial CFD codes solving RANS equations are now widely used for the estimation of propeller performance. It is possible to simulate the propeller flow not only for open
water condition but also with modelling the interaction between the propeller and the hull and even the rudder. An extensive research work [6] was focused on the coupling of blade element momentum theory with RANS method for an autonomous underwater vehicle. Uncertainty analysis has been carried out for the grid structure. Chase [7] has studied the self-propulsion performances of the DARPA suboff. A custom developed CFD solver has been employed for open water analyses at various advance coefficients. The effects of the turbulence models on the results have been investigated. The wake field has been compared with the experimental data for a constant advance coefficient [7]. A seven bladed INSEAN E1619 model propeller has been studied in the presence of DARPA suboff submarine model in [8]. The numerical analyses have been made by employing delayed detached eddy simulation approach. The results have been compared with different turbulence models using four grids and three time steps for one advance coefficient. The results showed that the present approach is applicable in self-propulsion performance prediction for submarines.

A study involving the interaction between propeller and a submarine hull was carried out in [9]. The analyses have been made by taking the free surface effects into account. The results showed a good agreement with the experiments. It has been highlighted that the free surface has significant effect on total resistance. The effects of bow and stern geometries on resistance of bared DARPA suboff via CFD method has thoroughly been studied in [10]. In another study [11], the hull-propeller interaction of DARPA suboff vehicle has been investigated by using body force method. Hydrodynamic performance of detached eddy simulations or large eddy simulations (LES hereafter) methods.

URANS-based methods suffer because the averaging of the main physical parameters cannot afford a proper appreciation of the intrinsic characteristics of the flow whereas the LES method, although more accurate, remains too expensive for common applications since it requires a highly accurate discretization in space and time. In between, there is the DES choice, which is applied herein. When the turbulent length scale exceeds the grid dimension, the model switches to a subgrid scale formulation and the flow is solved using the LES model. The DES approach is based on an explicit splitting of the computational domain into two zones. In the region near solid walls, the conventional RANS equations are solved. Within the second region, the governing equations are the filtered Navier Stokes equations of the LES approach. Since the hybrid nature of DES is not linked to any specific turbulence model, the one employed in here is a variant based on the k-ω SST model.

Continuing the scientific interest of the work group to which the author belongs, which regards either the seakeeping problem [12], the shallow water navigation [13] or the planning regime [14], the present paper deals with another atypical problem, namely the turbulent flow around a fully appended submerged body. It therefore describes a numerical investigation of the flow around the DARPA suboff appended hull at several velocities aimed at determine accurately the hydrodynamic resistance of the fully immersed hull. The axisymmetric hull is composed of a bow forebody, a cylindrical middle body section, and a curved stern. The hull has a maximum diameter and a length which is 8.6 times the diameter. The appendages raise particular challenges not only in handling with the complex geometric boundaries, but also for capturing the flow features such as boundary layer, junction flows, tip flows, and their interactions, which provide a sufficient complex model for investigating the robustness of the flow solver. Therefore, the significance of the reported work consists in the particular treatments of the otherwise classical formulations of the initial and boundary flow conditions used for a common surface ship. For this general class of computational applications there is no accepted best practice despite of its increasing interest shown in the applied ship hydrodynamics.

2. Geometry, computational domain and mesh generation
Since the present work deals with a flow problem without free-surface the minimal requirements imposed for the computational domain that hosts this type of flow are not applicable anymore. The only restriction that applies concerns the distance between the hull and the top and bottom boundaries, see figure 1, so that the suboff influence on these frontiers should remain at a less significant level. The main particulars of the suboff hull are tabulated in table 1 in which the hull length is considered as the
length between perpendiculars \( L_{PP} \). The hull consists of a revolution body, a sail placed at about 0.25 \( L_{PP} \), pairs of fins and rudders placed at the aft part of the body. The overall dimensions of the computational domain are as follows: the inlet boundary is located at 2\( L_{PP} \) in front of the hull, whereas the outlet is located at 4 \( L_{PP} \) behind the hull. Lateral frontiers are located 2\( L_{PP} \) on each side as the top and bottom boundaries are placed.

**Table 1. Suboff main particulars.**

| Parameter            | Value |
|----------------------|-------|
| Hull length [m]      | 4.356 |
| Hull diameter [m]    | 0.508 |
| Sails                | 1     |
| Fins                 | 2     |
| Rudders              | 2     |

![Figure 1. Computational domain.](image)

Whenever a viscous flow is numerically computed a sufficient number of grid points inside the boundary layer is required. A local Reynolds number based on the wall variable \( y^+ \) is computed prior to the grid generation, to estimate an appropriate cell size \( y_{wall} \) for the RANS solution. In the present study the mesh is generated by using the Hexpress module of the FINE\textsuperscript{TM}/Marine in which automatic refinement based on defined sensors either next to solid walls or inside specified area in the domain is possible. Special attention to the quality criteria such as orthogonality, smoothness and clustering is paid during the generation process. Based on the findings of previous works of the author [15, 16], several conditions concerning the cell clustering are imposed inside the boundary layers around the hull and all the appendages inside all the intersections of different surfaces, as figures 2 and 3 bear out. Figure 2 shows the mesh at the most important parts of the hull, i.e. extremities and the junction between the sail and hull. Similarly, figure 3 depicts the grid around intersection between the hull, fins, rudders and sail.

3. **Numerical approach**

The flow around the fully submerged hull is computed by using the ISIS-CFD solver of the Numeca FineTM/Marine package, which is employing the finite volume method. The solver uses algorithms providing a strong pressure-velocity coupling for the RANS equations. The simulation is accomplished in a global approach in which the momentum and mass conservation equations written in respect to a Cartesian system of coordinates are solved. Since no coordinate transformation is done in the solving algorithm, the efficiency of the numerical approach may be considered as being suited to the purpose. Dependent variables of the set of equations are the velocity and pressure. Closure to the turbulence is achieved either through the EASM model of Rumsey and Gatski [17], or by using the DES model, which is based on an implicit splitting of the computational domain into two zones. Inside the regions adjacent to the ship hull and wherever the turbulent length scale is less than the maximum grid dimension, the flow is solved based on the \( K-\omega \) SST model. On the contrary, when the turbulent length scale exceeds the grid dimension, the model switches to a subgrid scale formulation and the flow is solved using the LES model. The DES approach is based on an implicit splitting of the computational domain into two zones. In the first region near solid walls, the conventional RANS equations are solved. Within the second region, the governing equations are the filtered Navier-Stokes equations of the LES approach. The integration of the forces is performed on the solid-surface cell based on the quaternions formulation. Integration in time is done in an Euler explicit way, whereas an upwind discretization scheme is used for the convective terms with a second order for the acceleration. Conservation applies to the mass and momentum and a Piccard model applies for the linearization.
The pressure-correction is imposed and the Krylov technique is used for the iteration of the solution. A non-structured grid is used for the discretization of the computational domain, therefore hexahedral elements are used for that purpose. A fully unsteady approach is employed to advance the solution in time.

![Figure 2](image1.png)

**Figure 2.** Computational grid at: (a)-aft (b)-midship and (c)-fore of the subof hull.

![Figure 3](image2.png)

**Figure 3.** Details of the computational grid around the intersections of the hull with appendices.

### 3.1 Boundary condition formulations
The particularity of this type of flow resides in the absence of the free surface therefore the mobile frontier of the computational domain has to be replaced by an infinite one at the top of it, see figure 1. To avoid any unwanted influence of the submerged hull on that boundary, a mirror-type condition is used there. Similarly, the same type of condition is imposed on the bottom boundary. On the lateral frontiers the far-field condition is imposed. At the up-stream the flow is accelerated from the rest to the incoming flow velocity over a period of five seconds physical time. The hydrostatic frozen pressure is imposed at the downstream whereas the wall function is used on the solid boundary of the hull.

### 4. Results and discussions
Three grids were generated for performing the grid convergence test at first. They are denoted by G1…G3 in table 2. The associated number of cells was gradually increased, as shown in table 2, which tabulates the associated computed hydrodynamic resistance for a series of five velocities versus the experimental one reported in [18]. For the sake of consistency, the comparison will consider the solution computed only based on the EASM turbulence model. Although all the meshes have a rather low number of cells, the level of the errors, which is less than 4.22% for the coarsest grid and the
highest velocity, may be considered as being satisfactory at least for quick estimations of the resistance, as depicted in figure 4. Worth to underline that in the finest grid case, the errors do not overpass the 2% value. The time history of the numerical solution depicted in figure 5 reveals that rather soon after achieving the hull speed, the convergence is attained within a couple of seconds. The fast convergence is possible only because the flow is without a free-surface, therefore the wave fluctuations, otherwise significantly influential on the numerics, do not affect any more the stability of the solution. Given the results of the grid convergence test, all the following computations will be performed on the grid G3.

| Grid | $R_T$ [N] | $v=6kn$ | $v=9kn$ | $v=12kn$ | $v=15kn$ | $v=18kn$ |
|------|-----------|---------|---------|---------|---------|---------|
| EFD  | 105.24    | 239.37  | 389.83  | 605.35  | 837.71  |
| G1 (3.62 M) CFD | 101.107 | 235.311 | 400.547 | 614.22 | 802.36 |
| $\varepsilon$ [%] | 3.93 | 1.70 | 2.75 | 1.47 | 4.22 |
| G2 (6.31 M) CFD | 102.010 | 236.851 | 395.42 | 610.81 | 812.812 |
| $\varepsilon$ [%] | 3.07 | 1.05 | 1.43 | 2.97 |
| G3 (12.58 M) CFD | 103.128 | 239.267 | 393.416 | 606.787 | 828.765 |
| $\varepsilon$ [%] | 2.00 | 0.04 | 0.92 | 0.9 | 1.08 |

**Table 2.** Grid convergence test.

![Figure 4. Grid convergence test.](image1)

![Figure 5. Time history of the numerical solution.](image2)

Next, a comparative analysis of the flow structures around the hull is proposed for clarifying the differences between the EASM and DES turbulence models in terms of the capacity of reproducing the intrinsic details of the local flow. For this purpose, a comparison between the iso-surfaces computed for the second invariant of velocity tensor is shown in figure 6. The iso-surfaces of the second invariant equals 50 are drawn for the solutions based on the EASM model in figure 7(a) and DES model in figure 7(b). They both are coloured in terms of the helicity. Areas of rather intense change of the hydrodynamic parameters are seen on the suboff extremities and in the free stream of it, as expected. Although more expensive in terms of the CPU time, the DES solution seems to develop better in the wake of the appended hull. This is an important issue that may have a significant influence when the self-propulsion will be numerically simulated since the nominal wake has an overwhelming influence on the propeller performances. The cores of the paired vortices generated by the tips of the sail and fins as well as by their roots are perfectly anti-symmetric, a fact which proves again the overall accuracy of the ISIS-CFD solver used in the present study. Although not as intensive
as the tip-generated vortices, the horseshoe vertices provoked by the adverse pressure gradient at the intersection between the hull and the appendages have to be taken into account.

Within the boundary layer developing along a solid surface, the velocity profile exhibits a significant defect. Such a defect works as a trigger for the transition to turbulence and may even be presumed to be a sustaining source for the turbulent state in such flows. When an oncoming flow encounters a piercing body, the flow may separate as a result of a sort of blocking effect. The separated flow forms several vortices around the body, which are topologically rather similar to the horseshoe vortices generated around a juncture. The flow is completely three-dimensional and becomes more complicated as the vortices interact with the boundary layer developed on the body surface. This convoluted three-dimensional flow can be found in many places, such as the flow around appendages, antennas or periscopes mounted on submarine fuselages, etc.

![Figure 6. Comparison between the isosurfaces of the second invariant computed for \( v=6\) kn by using: up - the EASM turbulence model; down - the DES model.](image)

Such complex flow can affect the lift and stability/control characteristics of the appendages through the generation of horseshoe vortices. In spite of its engineering importance, there is no established method for estimating the wave resistance or wake characteristics because the detailed flow mechanism is not completely understood. Although some studies have been performed either theoretically or through conventional tank tests, the estimation of the wave resistance under the breaking assumption is hampered by the uncertainty associated with the complexity of the phenomenon. The heterogeneous character of the flow is due to the number of vortices originating upstream of the piercing body. These vortices result from the wave-induced separation due to the adverse pressure gradient in front of the body. Complex interactions that produce there unveil topological flow structures that resemble to the classical ones generated by the contingency between a separated flow around a juncture and the corresponding boundary layer. The phenomenon was extensively studied in [15, 16] and [19] where the authors were able not only to discuss the flow conditions and uncertainty analysis, but also to derive and verify a new topological rule for a surface-piercing body configuration.

An insight into the local flow features around the hull and right behind of it is proposed in the followings. Figure 7 bears out a comparison between the non-dimensional pressure fields computed for (a) – \( v=6\) kn, (b) – \( v=9\) kn, (c) – \( v=12\) kn and (d) – \( v=15\) kn, respectively. The pressure variation is
strongly dependent on the velocity, as expected. Areas of high pressure are present not only around the nose of the suboff bow, but also in front of each appendage, being located along the lines where the stagnation points lay. On the other hand, regions of minimum pressure are detected around the lateral faces of the sail and fins as well as on the hull inside the region where its diameter begins to decrease, a fact which respects the physics of the flow. Obviously, the quick change in the pressure field together with the vortices released by the sail and the aft appendages will affect the wake structure, therefore it will eventually affect the propeller working conditions. Aside of that, the presence of the fins at the aft region will also contribute to the distortion of the velocity field in spite of the fact that the hull is a body of revolution and one may expect a uniform wake structure. Even though the propeller is a seven-bladed one, the aforementioned distortion is expected to affect not only the propeller working regime, in terms of the bearing forces, but also the hydroacoustic pattern of the suboff, with severe consequences if the ship is used for strategic purposes. The above statement is sustained by figure 8, which bears out the streamwise velocity contours computed for $v=6\text{kn}$ (left figure) and $v=18\text{kn}$ (right figure). The distortion of the velocity field can be more clearly emphasized in a detailed representation of the wake.

![Figure 7](image1.png)
![Figure 8](image2.png)

**Figure 7.** Comparison between the pressure contours on the hull. (a) $v=6\text{kn}$, (b) $v=9\text{kn}$, (c) $v=12\text{kn}$, (d) $v=15\text{kn}$.

**Figure 8.** Comparison between the streamwise velocity fields on the hull and behind of it. Left: $v=6\text{kn}$, right: $v=18\text{kn}$.
A close-up view of the streamwise velocity fields drawn at 0.04L_{PP} behind the hull for the extreme speeds, namely v=6kn and v=18kn are shown in figure 9, which reveals that the velocity defect in the wake gets larger as the suboff speed increases. A similar conclusion was withdrawn during the experiments reported in [20]. Since the wake influence on the overall performances of the propeller may be significant, some additional work may be required either for reshaping the hull forms down there or for placing an appropriate appendage meant to smooth the flow such as a ring which can be mounted close to the fins and rudders at the aft part of the hull of the suboff.

Figure 9. Close-up view of the streamwise velocity contours at 0.04L_{PP} behind the hull. Left: v=6kn, right: v=18kn.

5. Conclusions
The paper describes a thorough investigation of the hydrodynamic performances of the DARPA suboff hull. The flow is numerically simulated by integrating in time the unsteady Navier-Stokes equations in which closure to the turbulence is achieved mainly by means of a modified DES approach. The solver used to achieve the purpose is the ISIS-CFD, part of the Numeca Fine^{TM}/Marine suite, which employs the finite volume of fluid technique. All the computations were performed on 120 processors of the HPC existent at the “Dunarea de Jos” of Galati. A grid convergence test was performed at first and the computed solutions proved not only a satisfactory accuracy of the solver, but also a rather fast convergence of the solution. The flow has been studied for five different incoming velocities and the comparisons with the available experimental data validate the numerical solutions. Based on the discussions provided in the preceding sections, the following findings may conclude the present study:
- Whenever the intrinsic details of the flow are of a primary interest, the detached eddy simulation is recommended despite of its higher costs;
- Although the study regarded a body of revolution, the computed wake reveals significant velocity defects, a fact that may affect negatively the propeller performances;
- An additional appendage is probably required at the aft of the suboff hull to smooth the flow in the wake;
- Based on the methodology developed in [21] the next step of the present research will regard the self-propulsion problem and eventually the 6DOF navigation close to the wavy free-surface following the model developed in [22].

6. References
[1] Bandyopadhyay P R 2005 Trends in biorobotic autonomous undersea vehicles IEEE J. Ocean. Eng. 30 109-139
[2] Wu X C, Wang, Y W, Huang C G et al 2015 An effective CFD approach for marine-vehicle maneuvering simulation based on the hybrid reference frames method Ocean Eng. 109 83-92
[3] Yang Y and Pullin D I 2011 Evolution of vortex-surface fields in viscous Taylor-Green and Kida-Pelz flows J. Fluid Mech. 685 146-164

[4] Zhao Y M, Yang Y and Chen S Y 2016 Vortex reconnection in the late transition in channel flow J. Fluid Mech. 802 R4

[5] Vaz G, Toxopeus S and Holmes S 2010 Calculation of manoeuvring forces on submarines using two viscous-flow solvers Proc. of ASME 29th International Conference on Ocean, Offshore and Arctic Engineering OMAE (Shanghai, China) 6 pp 621-633

[6] Philips A B, Turnock S R and Furlong M 2008 Comparisons of CFD simulations and in-service data for the self-propelled performance of an autonomous underwater vehicle Proc. of 27th Symposium on Naval Hydrodynamics (Korea)

[7] Chase N 2012 Simulations of the DARPA Suboff Submarine Including Self-propulsion with the E1619 Propeller Master Thesis University of Iowa

[8] Chase N and Carrica P M 2013 Submarine propeller computations and application to self-propulsion of DARPA Suboff Ocean Eng. 60 68-80

[9] Zhang N and Zhang S 2014 Numerical simulation of hull/propeller interaction of submarine in submergence and near surface conditions J. Hydrodyn. Ser. B 26 50-56

[10] Budak G and Beji S 2016 Computational resistance analyses of a generic submarine hull form and its geometric variants J. Sci. Technol. 11 77-86

[11] Delen C, Sezen S and Bal S 2017 Computational investigation of self-propulsion performance of DARPA Suboff vehicle TAMAP J. Eng. 1-12

[12] Pacuraru F and Domnisoru L 2017 Numerical investigation of shallow water effect on a barge ship resistance IOP Conf. Ser.: Mat. Sci. and Eng. 227

[13] Pacuraru S, Domnisoru L and Pacuraru F 2018 Numerical study on motions of a containership on head waves IOP Conf. Ser. Mat. Sci. and Eng. 400(8)

[14] Caramatescu A, Pacuraru F and Cristea C 2016 Simulation of a cargo planning boat with inverted keel, Proc. of the International Conference on Traffic and Transport Engineering, (Belgrade, Serbia) pp 265-271

[15] Lungu A and Ungureanu A 2008 Numerical study of a 3-D juncture flow AIP Conf. Proc. 1048(1) pp 839-842

[16] Lungu A and Ungureanu C 2009 Numerical simulation of the turbulent flow around a strut mounted on a plate AIP Conference Proc. 1168(1) pp 647-650

[17] Rumsey C L and Gatski T B 2001 Recent turbulence model advances applied to multielement airfoil computations J. of Aircraft 38(5) 904–910

[18] Liu H L and Huang TT 1998 Summary of DARPA suboff experimental program data (No. CRDKNSWC/HD-1298-11) Naval Surface Warfare Center Carderock Division (NSWCCD) West Bethesda MD USA

[19] Zhang Z J and Stern F 1996 Free-surface wave-induced separation J. of Fluids Engineering 118 546-554

[20] Crook B 1990 Resistance for DARPA suboff as represented by model 5470 David Taylor Research Center report DTRC/SHD-1298-07

[21] Bekhit A S 2018 Numerical simulation of the ship self-propulsion prediction using body force method and fully discretized propeller model IOP Conf. Ser.: Mat. Sci. and Eng. 400

[22] Bekhit A S 2018 Unsteady RANSE simulation for ship resistance, heave and pitch in regular head waves IOP Conf. Ser.: Mat. Sci. and Eng. 400