SIMULATION OF SUBMARINE MANOEUVRING USING NAVIER-STOKES SOLVER

A. DROUET*, G. RELIQUET*, A. BARDIN*, E. JACQUIN*, L. GENTAZ† AND B. ALESSANDRINI†

* HydrOcean
1 rue de la Noë, CS 32122, 44321 Nantes Cedex 3, France
E-mail: aurelien.drouet@hydrocean.fr, web page: http://www.hydrocean.fr

† Hydrodynamics and Ocean Engineering Group, Fluid Mechanics Lab
Ecole Centrale Nantes, 1 rue de la Noë, 44300 Nantes, France

Key words: Manoeuvring, Multi-blocks, Navier-Stokes Solver, Finite Differences, Submarine

Summary. (optional)

1. INTRODUCTION

The prediction of manoeuvring of underwater vehicles such as UAV or submarines is a key point for their performance and safety. From an experimental point of view, such manoeuvring characteristics can be determined either by captive model tests coupled with a mathematical model or by free model tests. The first method consists in determining linear and non-linear hydrodynamic coefficients from fixed cases (combination of drift or incidence angles, gyrating radius in vertical or horizontal plane, for several appendages angles). It requires a very large number of tests to be efficient. The second method needs more complicated models (autonomous, underwater measurements ...), and very large facilities in which manoeuvres can be performed. Therefore, naval architects and engineers in charge of designing underwater vehicles are interested in having at their disposal accurate numerical alternatives. HydrOcean and Ecole Centrale Nantes are involved in a cooperative work in order to develop and validate numerical solvers and associated methodologies in order to accurately compute submarines manoeuvring performances with acceptable CPU time.

The final objective is to perform fully free and self-propelled simulation. This requires a RANSE free surface solver able to accurately predict forces and moments applied along the hull of a submerged body and on its appendages. In that way, this paper presents validations that have been conducted on a fixed model of the DARPA Suboff model for several angles of drift, and different configurations.

2. GENERAL DESCRIPTION OF ICARE SOLVER

ICARE [5],[6] is a RANSE (Reynolds Average Navier-Stokes Equations) free-surface solver initially co-developed by Ecole Centrale Nantes under French Ministry of Defense support, and by HydrOcean. It uses the k-ω turbulence model developed by Wilcox, 1988. General schemes are based on second order (in space and time) implicit finite differences. Discrete unknowns are distributed on a hexahedral structured curvilinear grid fitted to the hull and the
free surface. Recently the solver has been modified in order to take into account multi-block grids therefore this new version model has to be validated. The picture below show a cylinder meshed using two topology methods: Fully H-H and multi-blocks O-grid.

![Example of H-H grid topology (left) and multi-blocks O-grid (right)](image)

The use of O-grid structure reduces the grid size because refinements are concentrated near walls and increases grid quality.

The reading, treating and writing of the variables were adapted to handle the multi-block structure of the grid. Implemented modifications are quite similar to those of a parallel solver with each block treated by a single processor using the MPI library. Each block has its own data and the same structure is used in all blocks. Specific procedures have been implemented to link blocks and to transfer information through block interfaces. Information coming from surrounding blocks and required to compute a specific block are stored in specific areas and updated when surrounding blocks are computed. A sketch of the mapping where information is stored in surrounding blocks is presented in the following figure.

![block mapping](image)

The resolution is operated using a global method that computes the whole grid directly. Information is updated in the transfer area during the resolution. The linear system is exactly solved as in the single block version of the solver.
3. NUMERICAL SETUP

Geometry

The geometry is the DARPA SUBOFF submarine hull form because of the amount of data available for validating flow field or integral quantities [2],[3],[4]. The description of the different elements composing the geometry is described by Groves et al [1]. Computations have been conducted first on the bare hull and then on several configurations of the bare hull with fairwater or stern appendages. The mains characteristics of the DARPA SUBOFF submarine hull form are specified in the table below.

<table>
<thead>
<tr>
<th>Description</th>
<th>Symbol</th>
<th>Magnitude</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length overall</td>
<td>Loa</td>
<td>4.356</td>
<td>m</td>
</tr>
<tr>
<td>Length between perpendicular</td>
<td>Lpp</td>
<td>4.256</td>
<td>m</td>
</tr>
<tr>
<td>Maximum hull radius</td>
<td>Rmax</td>
<td>0.254</td>
<td>m</td>
</tr>
<tr>
<td>Longitudinal position of the center of buoyancy</td>
<td>Lcb</td>
<td>0.4621 Loa</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 1 : Design Parameters

The table below presents the three configurations that have been studied in this paper.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hull</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Sail</td>
<td></td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Stern Appendages</td>
<td></td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 2 : Description of the configurations

The considered plane of motion is horizontal.
The pictures below show views of the bare hull and of the different elements such as sail or stern appendages.

Figure 3 : View of the geometry of the DARPA SUBOFF bare hull, zoom on sail and stern appendages

Computational conditions

Two sets of experimental data based on two Reynolds numbers are available. The first set corresponding to a Reynolds number of 12 million gives us experimental data on the flow field around the bare hull at angles of drift of 0° and 2°. The second data set corresponding to a Reynolds number of 14 million gives us integral quantities function of the angle of drift from 0° to 18°. All computations have been performed with an inflow speed of 6.5 knots with two kinematic viscosities in order to adapt Reynolds number. Depending on the configuration, two planes of motions have been studied for angles of drift from 0° to 18°. The table below presents the computation matrix:
Definitions
The axis definition is the same as the one used in the experiments with $x$ directed aft, $y$ to starboard and $z$ vertically upward. The origin of the right-handed system of axes is located in the revolution axis of the bare hull at the longitudinal position of the LCB.

All coordinates given in this paper are made non-dimensional with the length overall $Loa$ of the submarine ($Loa = 4.356m$). The velocity field $V=(u,v,w)$ is made non-dimensional with the undisturbed velocity $V_0$. The angle of drift is defined by $\beta = \arctan \frac{u}{v}$ with $u$ and $v$ directed respectively according to $x$ and $y$ axes, which means that $\beta$ is positive for a flow coming from the port side.

All integral forces and moments applied on the hull are based on a right-handed system of axes corresponding to positive directions normally used in maneuvering studies. $X$ is directed forward, $Y$ to starboard and $Z$ downward. According to results presented by Roddy [3], all moment are expressed at center of gravity, which is located at $0.4621 \text{ Loa}$ aft of the nose of the submarine. Non-dimensionalisation is done with the length between perpendicular ($Lpp = 4.621m$) using $X, Y, Z/\frac{1}{2} \rho V_o^2 L_{pp}^2$ and $K, M, N/\frac{1}{2} \rho V_o^2 L_{pp}^3$. The pressure coefficient $C_p$ is defined as $C_p = (p - p_0)/\frac{1}{2} \rho V_o^2$, $p_0$ being the undisturbed pressure.

Meshing strategy
The computational domain is discretized using structured hexahedral O-grid meshes created with IcemCFD meshing software. Finer cells are used to discretize the boundary layer properly near the solid walls. The non-dimensional first distance from the wall is $10^{-4}$. The figure below shows views of the mesh around the bare hull and appendages.

![Figure 4: Meshing details around the hull and appendages](image)

4. CONVERGENCE STUDY
A convergence study has been conducted on the bare hull geometry. The influence of the grid density has been studied separately on the nose and on the stern of the submarine. The table below shows the different longitudinal numbers of nodes on the different parts.
The axis definition is the same as the one used in the experiments with $x$ directed aft, $y$ to starboard and $z$ vertically upward. The origin of the right-handed system of axes is located in the revolution axis of the bare hull at the longitudinal position of the LCB. All coordinates given in this paper are made non-dimensional with the length overall $Loa$ of the submarine ($Loa = 4.356m$). The velocity field $V=(u,v,w)$ is made non-dimensional with the undisturbed velocity $V_0$. The angle of drift is defined by with $u$ and $v$ directed respectively according to $x$ and $y$ axes, which means that $\angle$ is positive for a flow coming from the port side.

All integral forces and moments applied on the hull are based on a right handed system of axes corresponding to positive directions normally used in maneuvering studies. $X$ is directed forward, $Y$ to starboard and $Z$ downward. According to results presented by Roddy [3], all moment are expressed at center of gravity, which is located at $0.4621 Loa$ aft of the nose of the submarine. Non-dimensionalisation is done with the length between perpendicular ($Lpp = 4.621m$) using and . The pressure coefficient is defined as $( )$, being the undisturbed pressure.

### Meshing strategy

The computational domain is discretized using structured hexahedral O-grid meshes created with IcemCFD meshing software. Finer cells are used to discretize the boundary layer properly near the solid walls. The non-dimensional first distance from the wall is $10^{-4}$. The figure below shows views of the mesh around the bare hull and appendages.

![Figure 4: Meshing details around the hull and appendages](image)

### 4. CONVERGENCE STUDY

A convergence study has been conducted on the bare hull geometry. The influence of the grid density has been studied separately on the nose and on the stern of the submarine. The table below shows the different longitudinal numbers of nodes on the different parts.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Bow</th>
<th>Middle</th>
<th>Stern</th>
<th>Mesh size</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>60</td>
<td>60</td>
<td>110</td>
<td>850 000</td>
</tr>
<tr>
<td>2</td>
<td>60</td>
<td>32</td>
<td>110</td>
<td>760 000</td>
</tr>
<tr>
<td>3</td>
<td>40</td>
<td>32</td>
<td>110</td>
<td>700 000</td>
</tr>
<tr>
<td>4</td>
<td>25</td>
<td>32</td>
<td>110</td>
<td>650 000</td>
</tr>
<tr>
<td>5</td>
<td>60</td>
<td>32</td>
<td>80</td>
<td>665 000</td>
</tr>
<tr>
<td>6</td>
<td>60</td>
<td>32</td>
<td>60</td>
<td>600 000</td>
</tr>
<tr>
<td>7</td>
<td>60</td>
<td>32</td>
<td>35</td>
<td>520 000</td>
</tr>
</tbody>
</table>

Table 4: Characteristics of the grids used in the convergence study

The meshes 1 and 2 have been used as references for the convergence study. Meshes 2, 3 and 4 are compared for bow mesh density study and meshes 2, 5, 6, 7 are compared to the stern study. The convergence study has been conducted for a velocity of 6.5 knots which corresponds to a Reynolds number (based on the length between perpendiculars) of about 14 million combined with three angles of drift $0^\circ$, $8^\circ$ and $18^\circ$.

The table below shows the differences between results of drag and lift obtained with the mesh 2 in comparison with those obtained with mesh 1. The decomposition of the total force in terms of pressure and friction parts is given below.

<table>
<thead>
<tr>
<th>Differences in comparison with mesh 1 (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angles of drift (°)</td>
</tr>
<tr>
<td>0</td>
</tr>
<tr>
<td>8</td>
</tr>
<tr>
<td>18</td>
</tr>
</tbody>
</table>

Table 5: Differences between results of mesh 2 in comparison with mesh 1

The reduction of the number of nodes in the parallel mid-body has no impact on the forces and their decomposition for the three angles of drift.

**Mesh density on the nose**

The figures below present the three grid densities that have been tested on the nose of the submarine.

![Figure 5: Grids with varied densities on the nose](image)
The table below shows the differences obtained by comparing results obtained with meshes 3 and 4 to results obtained with mesh 2.

![Table 6: Differences between results of mesh 3 and 4 in comparison with mesh 2](image)

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Angles of drift (°)</th>
<th>Xp</th>
<th>Xf</th>
<th>X</th>
<th>Yp</th>
<th>Yf</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0</td>
<td>-0.56%</td>
<td>0.23%</td>
<td>0.14%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>-0.16%</td>
<td>0.13%</td>
<td>0.07%</td>
<td>0.06%</td>
<td>0.10%</td>
<td>0.06%</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>-0.07%</td>
<td>0.13%</td>
<td>0.01%</td>
<td>-0.05%</td>
<td>-0.08%</td>
<td>-0.05%</td>
</tr>
<tr>
<td>4</td>
<td>0</td>
<td>-1.36%</td>
<td>0.59%</td>
<td>0.38%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>-0.30%</td>
<td>0.50%</td>
<td>0.33%</td>
<td>0.16%</td>
<td>0.69%</td>
<td>0.15%</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>-0.22%</td>
<td>0.34%</td>
<td>0.00%</td>
<td>-0.25%</td>
<td>-0.66%</td>
<td>-0.24%</td>
</tr>
</tbody>
</table>

The biggest differences are obtained for the pressure drag at angle of drift of 0° for the two meshes with respectively -0.56% for the mesh 3 and -1.36% for the mesh 4. The differences between meshes 4 and 2 are relatively small, justifying the use of the mesh 4 for the follow-up of the study.

**Mesh density on the stern**

The figures below show the four grid densities that have been tested on the stern of the submarine.

![Grid 2](image)  ![Grid 5](image)

![Grid 6](image)  ![Grid 7](image)

**Figure 6: Grids with varied density on the stern**

The table below shows the differences obtained by comparing results from meshes 5, 6 and 7 to results from mesh 2.
Table 6: Differences between results of mesh 3 and 4 in comparison with mesh 2

The biggest differences are obtained for the pressure drag at angle of drift of 0° for the two meshes with respectively -0.56% for the mesh 3 and -1.36% for the mesh 4. The differences between meshes 4 and 2 are relatively small, justifying the use of the mesh 4 for the follow-up of the study.

Mesh density on the stern

The figures below show the four grid densities that have been tested on the stern of the submarine.

![Figure 6: Grids with varied density on the stern](image)

The table below shows the differences obtained by comparing results from meshes 5, 6 and 7 to results from mesh 2.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Angles of drift (°)</th>
<th>Xp</th>
<th>Xf</th>
<th>X</th>
<th>Yp</th>
<th>Yf</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>1.07%</td>
<td>-0.03%</td>
<td>0.09%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.24%</td>
<td>-0.01%</td>
<td>0.04%</td>
<td>-0.24%</td>
<td>-0.60%</td>
<td>-0.23%</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>-0.13%</td>
<td>-0.01%</td>
<td>-0.08%</td>
<td>-0.12%</td>
<td>-0.05%</td>
<td>-0.12%</td>
</tr>
<tr>
<td>6</td>
<td>0</td>
<td>3.88%</td>
<td>-0.07%</td>
<td>0.36%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>1.05%</td>
<td>-0.05%</td>
<td>0.19%</td>
<td>-0.64%</td>
<td>-1.55%</td>
<td>-0.63%</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>-0.30%</td>
<td>-0.04%</td>
<td>-0.20%</td>
<td>-0.31%</td>
<td>-0.18%</td>
<td>-0.31%</td>
</tr>
<tr>
<td>7</td>
<td>0</td>
<td>10.61%</td>
<td>-0.26%</td>
<td>2.08%</td>
<td>-1.27%</td>
<td>-2.25%</td>
<td>-1.26%</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>28.05%</td>
<td>-0.27%</td>
<td>2.81%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>0.29%</td>
<td>-0.28%</td>
<td>0.07%</td>
<td>-0.71%</td>
<td>-0.29%</td>
<td>-0.71%</td>
</tr>
</tbody>
</table>

Table 7: Differences between results of mesh 5, 6 and 7 in comparison with mesh 2

The results obtained with mesh 6 and 7 are not close to those obtained with mesh 2 since the differences in drag is about 4% for mesh 6 and 28% for mesh 7. The results obtained with mesh 5 are more satisfying since the differences with results obtained using mesh 2 is about 1%, that is acceptable. The mesh 5 will be retained as the better grid refinement on the stern.

Final mesh

The final mesh is a combination of the grid density on the nose of mesh 4 with grid density of the stern of mesh 5. The table below shows the differences obtained by comparing results from the final mesh to results from mesh 2.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Angles of drift (°)</th>
<th>Xp</th>
<th>Xf</th>
<th>X</th>
<th>Yp</th>
<th>Yf</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>-0.28%</td>
<td>0.56%</td>
<td>0.47%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>-0.06%</td>
<td>0.48%</td>
<td>0.37%</td>
<td>-0.08%</td>
<td>0.08%</td>
<td>-0.09%</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>-0.35%</td>
<td>0.34%</td>
<td>-0.08%</td>
<td>-0.36%</td>
<td>-0.71%</td>
<td>-0.36%</td>
</tr>
</tbody>
</table>

Table 8: Differences between results of final mesh in comparison with mesh 2

As expected, the largest differences are about 0.5%. This mesh will be used for the validation study.
5. VALIDATIONS

Bare Hull
The grid used for the computations on the bare hull is the final mesh of the convergence study which is a structured hexahedral grid of about 525 000 cells for a half domain. Non-dimensional forces and moments applied on the bare hull for angles of drift from 0° to 18° are presented on the figure 7 with experimental data.

Figure 7: Non-dimensionalised forces (X’, Y’) and moments (N’) on the bare hull, computations (line) and experiments (dots)

Forces and moments coefficients are close to experimental data with higher differences (about 7%) for the difference between drag coefficients comparing computations and experiments. More investigations on the local quantities such as pressure coefficients, skin friction coefficients or flow field around the submarine have been conducted in comparison with experimental data. In the graph below the continuous line represents the pressure coefficients obtained by computations whilst those obtained experimentally are presented by dots.

Figure 8: Pressure (left) and one thousand times skin friction (right) coefficient along the hull, straight flight, computation (line), experiments (dots)
It can be seen that the pressure coefficient is well predicted and that there are greater differences between computations and experiments on the skin friction coefficient especially along the stern. The picture below shows the axial and radial velocity in the wake plane at X/Lpp=0.978.

![Figure 9: Axial and radial velocity, straight flight, computation (line), experiments (dots)](image)

It can be seen in the graph above that the flow pattern very close to the hull is quite well predicted with some differences due to turbulence modeling.

**Bare Hull + Sail**

This mesh was created introducing in the bare hull mesh the sail element through a new O-grid zone. This grid is constituted of about 890,000 nodes. Non-dimensional forces and moments applied on the bare hull + Sail for angles of drift from 0° to 18° are presented on the figure 10 with experimental data.

![Figure 10: Non-dimensionalised forces (X', Y') and moments (N') on the bare hull + Sail, computations (line) and experiments (dots)](image)
Figure 10 shows that forces and moments are well predicted up to a drift angle of 12° where flow separation occurs in computations and not in the experiments, where it occurs for an angle of drift of 16°.

**Bare Hull + Stern Appendages**

The grid has been created from the bare hull mesh where new O-grid structures have been added around stern appendages. The final grid is composed of about 1 million nodes.

Non-dimensional forces and moments applied on the bare hull + Sail for angles of drift from 0° to 18° are presented on the figure 11 with experimental data.

As for the bare hull more differences were obtained on the drag force than on lift force and moment, however forces and moments are globally well predicted.

The figure 12 presents a comparison of the mean axial velocity contours in the plan x/Lpp = 0.978 for the configuration bare hull, bare hull + sail and bare hull + stern appendages between computational results and experiments. Mean axial velocity contours seem to be well predicted for all configurations. The effect of the sail or of the stern appendages on the wake can be seen on the graph as it can be observed on the experimental results.
Figure 10 shows that forces and moments are well predicted up to a drift angle of 12° where flow separation occurs in computations and not in the experiments, where it occurs for an angle of drift of 16°.

Bare Hull + Stern Appendages

The grid has been created from the bare hull mesh where new O-grid structures have been added around stern appendages. The final grid is composed of about 1 million nodes.

Non-dimensional forces and moments applied on the bare hull + Sail for angles of drift from 0° to 18° are presented on the figure 11 with experimental data.

Figure 11: Non-dimensionalised forces \((X', Y')\) and moments \((N')\) on the bare hull + Stern Appendages, computations (line) and experiments (dots).

As for the bare hull more differences were obtained on the drag force than on lift force and moment, however forces and moments are globally well predicted.

The figure 12 presents a comparison of the mean axial velocity contours in the plan \(x/L_{pp} = 0.978\) for the configuration bare hull, bare hull + sail and bare hull + stern appendages between computational results and experiments. Mean axial velocity contours seem to be well predicted for all configurations. The effect of the sail or of the stern appendages on the wake can be seen on the graph as it can be observed on the experimental results.

The picture below shows the mean axial velocity contours for three angles of drift and for the three configurations.

Figure 12: Mean axial velocity contours at \(x/L_{pp} = 0.978\) for several configurations, Experiments (up), Computations (bottom).

The picture below shows the mean axial velocity contours for three angles of drift and for the three configurations.

Figure 13: \(V_x/V_0\) contours for several configurations.
Figure 13 first shows that the angle of drift is responsible for the creation of two counter-rotating vortices on the back end of the submarine that is present in the three configurations. The lift created by the drift angle on the sail or on the vertical stern appendages also initiates the creation of a vortex. The sail increases the intensity of the upper vortex and so we can see a dissymmetry in the wake. The effect of the sail and of the stern appendages on the flow pattern is clearly visible.

6. CONCLUSION

The latest release of our RANSE free surface solver Icare including Multi-blocs topologies has been evaluated for several configurations of the DARPA Suboff firstly on bare and secondly with appendages such as sail or stern appendages. The results obtained are generally good except for angles of drift larger than 12° for the bare hull + sail configuration. Such results could be explained by the choice of the turbulence model (k-ω by Wilcox).

The results obtained during the validation are very encouraging since the majority of the cases forces and moments applied on the submarine can be well predicted. Our next step will be the validation of gyration cases including fixed and fully free model.

REFERENCES