ABSTRACT: Hydrodynamic coefficients are the main significant factors that strongly affect the performance, controllability and maneuverability characteristics of autonomous underwater vehicles (AUVs). Experimental, numerical and empirical methods are different ways to predict hydrodynamic coefficients. Various captive tests, such as; straight-line towing, Rotating Arm (RA), Planar Motion Mechanism (PMM), and Coning Motion Mechanism (CMM) tests can be conducted to obtain hydrodynamic coefficients or maneuvering derivatives that are necessary for hydrodynamic design and system simulation of AUVs. Although the most reliable way to determine hydrodynamic coefficients of AUVs is conducting hydrodynamic tests, thanks to developments in computer technology it is possible to solve fluid problems and simulate captive model tests by using Computational Fluid Dynamics (CFD) methods.

In this study, CFD modelling techniques are developed to simulate straight-line towing tests, RA tests and PMM tests those are used in order to determine hydrodynamic characteristics and investigate hydrodynamic performance of AUVs. These underwater test mechanisms are modeled by using CFD methods. FLUENT commercial fluid solver is used to solve these fluid problems. UK Natural Environment Research Council’s Autosub test-case model and test results are used for CFD methods verification. Rotating arm tests are simulated for constant angular velocity and constant angular acceleration. Furthermore, pure heave motion is simulated in the scope of PMM studies. FLUENT user defined functions are used to implement defined dynamic motion of the body. Finally, analyses results are verified with experimental data which is available in literature and a new hydrodynamic simulation approach is developed for underwater vehicles.

Keyword: Hydrodynamic, PMM, RA, AUVs, Computational Fluid Dynamics

INTRODUCTION

In recent years, intensive efforts are being concerted towards the development of AUVs and AUVs have become a main tool for undersea survey for the scientific, military and commercial applications. Despite the considerable improvements in AUV usage area, AUV technologies are still attractive to scientist and engineers as a challenging field.

In the design stage of AUVs, it is necessary to analyze and determine its hydrodynamics performance, maneuverability and controllability characteristics. A useful approach to investigate the performance of an AUV is to apply system simulations and solve equations of the motion. To perform these simulations, the hydrodynamic coefficients of the vehicles have to be calculated. The hydrodynamic coefficients may be classified into two main groups. These are static hydrodynamic coefficients and dynamic hydrodynamic coefficients. Static hydrodynamic coefficients are used while the AUV have steady cruise conditions without any maneuvering. However, dynamic hydrodynamic coefficients are more complex than the statics, and dynamic coefficients can be classified into four subgroups such as; linear maneuvering coefficients, nonlinear maneuvering coefficients, coupled maneuvering
coefficients and added mass & inertia coefficients. These coefficients can be obtained by experimental, numerical and empirical methods however; the most reliable one is the experimental methods. Vertical Planar Motion Mechanism (V-PMM), Rotating-Arm Mechanism and Coning Motion Mechanism are the most common experimental techniques to measure the hydrodynamic coefficients. Rotating-Arm Mechanism and V-PMM are shown in Fig. 1.

![Figure 1-Rotating Arm Mechanism and V-PMM](image)

In spite of the fact that the most reliable methods to calculate hydrodynamic coefficients are the experimental techniques, also these are the most expensive and time consuming ones to determine the hydrodynamic coefficients. Because of this reason, experimental methods cannot be used in each step of the AUV design cycle. Instead of experimental methods, numerical (CFD) and empirical techniques can be used.

**DEFINITION OF THE HYDRODYNAMICS COEFFICIENTS**

Static and the linear maneuvering hydrodynamic coefficients mainly define the maneuverability of an AUV. A rectangular Cartesian coordinate system, attached to the center of gravity of vehicle, is used as a reference axis for the hydrodynamic coefficients. The three components of the hydrodynamic force along the directions x, y, z are denoted by X, Y, Z respectively, and the three components of the hydrodynamic moments by K, M, N. This is illustrated in Fig. 2.

![Figure 2-Schematic of the Cartesian coordinate system](image)

The path of the vehicle is then assumed to be intentionally altered slightly by deflection of various control surfaces on the vehicle. The three components of force X, Y, Z and the three components of the moments K, M, N are expanded up to second order terms in the linear
velocities u, v, w and the angular velocities p, q, r where these velocities now represent perturbations to the equilibrium condition of steady state forward motion. The expression for the forces and moments are derived from “Standard Equations of Motion for Submarine Simulations” [3] and then take the form:

\[ \sum X = X_{eq} q^2 + X_{rr} q r^2 + X_{rp} r p + X_{uu} u^2 + X_{uv} u v + X_{uv} v^2 + X_{ww} w^2 + X_{q\delta e} q \delta e^2 + X_{p\delta e} p \delta e^2 + X_{6\delta \delta e} \delta e^2 \]  

\[ \sum Y = Y_{eq} q + Y_{pp} p + Y_{pq} q r + Y_{qr} q r + Y_{uv} u v + Y_{uw} u w + Y_{u\delta e} u \delta e + Y_{v\delta e} v \delta e \]  

\[ \sum Z = Z_{eq} q + Z_{pp} p + Z_{pq} p q + Z_{qr} q r + Z_{uv} u v + Z_{uw} u w + Z_{q\delta e} q \delta e + Z_{p\delta e} p \delta e + Z_{6\delta \delta e} \delta e \]  

\[ \sum K = K_{pp} p + K_{pq} p q + K_{qr} q r + K_{uv} u v + K_{uw} u w + K_{q\delta e} q \delta e + K_{p\delta e} p \delta e + K_{6\delta \delta e} \delta e \]  

\[ \sum M = M_{eq} q + M_{pp} p + M_{qr} q r + M_{uv} u v + M_{uw} u w + M_{q\delta e} q \delta e + M_{p\delta e} p \delta e + M_{6\delta \delta e} \delta e \]  

\[ \sum N = N_{pp} p + N_{qr} q r + N_{uv} u v + N_{uw} u w + N_{q\delta e} q \delta e + N_{p\delta e} p \delta e + N_{6\delta \delta e} \delta e \]  

There are much kind of hydrodynamic coefficients which could be evaluated to describe the dynamics of the vehicle [4]. In this project static, linear & nonlinear maneuvering and added mass & inertia coefficients are calculated by using CFD simulations. Firstly, CFD analysis process and then verification studies will be investigated.

METHODOLOGY OF THE CFD ANALYSES PROCESS

In this report, a verification study has been performed for five degrees of freedom hydrodynamic coefficients such as surge, heave, sway forces and pitch, yaw moments. Roll moments verification study is planned as a future work. In order to verify simulation approach, linear and nonlinear CFD analyses are performed in steady and transient conditions.

Applied CFD analyses stages are listed as follow:

- Create geometry in CATIA V5 R21 and export in step file format Generate surface mesh using GAMBIT 2.4.1
- Generating Boundary Layer Region mesh in TGRID 2.4.1.
- Generate remaining volume mesh in GAMBIT 2.4.1 (between upper sides of the Boundary Layer to outer domain)
- Define Boundary Conditions and Fluid solver parameters in ANSYS FLUENT 13.0
- Check convergence and perform post-process.

Triangular elements on surfaces are used to generate wedge elements for boundary layer region for y+ value is 1. Pressure-based solver is used for the low-speed incompressible fluid. Steady-State CFD analyses are performed using different turbulence models which are Spalart-Allmaras, realizalbe k-ε and k-ω to investigate the effect of turbulence models on hydrodynamic coefficients.
SIMULATION MODEL AND VERIFICATION RESULTS OF HYDRODYNAMICS ANALYSIS

In this study, a comprehensive study has been made to verify the CFD tools for the hydrodynamic analysis of the underwater bodies. In order to evaluate the capabilities and accuracy of the tools used in this project, AUTOSUB autonomous underwater vehicle (AUV) model has been selected as a test case, because of the available geometric information, test conditions and extensive validation data. The experimental data of Autosub AUV used in the present study are based on the experiments which were performed in the National Oceanography Centre, Southampton.

Autosub Autonomous Underwater Vehicle Model

Autosub Project was initiated firstly in 1988, in order to develop unmanned autonomous underwater vehicles for its future marine science programs and for global monitoring at the National Oceanography Centre, Southampton [5]. Autosub is controlled by four movable control surface mounted at the rear of the vessel in a cruciform arrangement. The control surfaces’ cross sections consist of NACA 0015 hydrofoil [5]. Autosub’s principle dimensions and hydrofoil are listed below [5]:

- Length: 5.2 meter
- Diameter: 0.6685 meter
- Scale Factor: 0.743

Autosub model is composed of an axisymmetric body with four appendages in plus configuration. Picture and 3D drawing of the full model is given in Fig. 3.

Figure 3-Autosub AUV

In this study, three different type captive model test cases are simulated as in the follows:

- Straight-Line Towing Test Mechanism
- Rotating Arm Test (RA) Mechanism
- Planar Motion Mechanism (PMM)

Straight-Line Towing Test

Straight Line Towing test analyses are conducted to calculate the static coefficients and non-scaled model of AUV is used in the CFD analyses. AUTOSUB model is used for CFD analysis. Fig.4 and Fig.5 shows the surface mesh.
Applied boundary conditions are listed. Fluid domain is created as rectangular prisms and BCs:

- Side faces → symmetry
- Front face, lower face → velocity inlet
- Upper face, back face → pressure outlet
- Model surface → wall

Fluid domain and boundary conditions are shown in Fig. 6. CFD analyses are performed at 2.69 m/s velocity and for 0°, 2°, 4°, 6°, 8° and 10° angles of attack.

In the analyses, three different turbulence models are used to investigate the effect of turbulence model on hydrodynamic coefficients and comparison with experimental data is performed. CFD results of $Z'$ (hydrodynamic force coefficient in $z$-direction) and $M'$ (hydrodynamic moment coefficient in $y$-direction) are compared by experimental data in Fig.7 and Fig.8 [5].
As we see from above graph, best agreement for force in z direction and pitching moment is achieved by realizable k-ε model for \( Z' \) and \( M' \). Significantly, Spalart-Allmaras turbulence model which uses one equation to model turbulence could not capture viscous phenomena and calculated results deviate from the experimental data [5]. For the rest of the CFD analysis, realizable k-ε turbulence model will be used.

**Rotating Arm Test (RA) Mechanism**

Rotating Arm (RA) analysis is carried out for constant angular velocity by using AUTOSUB model. Only difference between RA grid and straight-line towing test grid is control volume with respect to shapes. Fluid domain of RA is modelled as semi-cylinder and constant angular velocity is defined for the fluid domain. Fig. 9 and Fig. 10 shows solution domain and boundary conditions

![Figure 9-Fluid Domain Grid for RA Test Simulations (Constant Angular Velocity)](image)

![Figure 10-Fluid Domain Boundary Conditions for Autosub RA Test Simulation (Constant Angular Velocity)](image)

CFD analyses are performed for cruise conditions indicated as follows:

- Turn Radius (R) [m] → 13, 17.385, 26
- Angular Velocity (q) [rad/s] → 2.69/R
- Ambient Pressure [Pa] → 150277

Firstly, force coefficient \((Z')\) in z-direction and moment coefficient \((M')\) in y-direction is found at indicated turn radius and then, q dependent dynamic coefficients \((Z'_q, M'_q)\) are calculated. Results are shown in Fig.11 and Fig.12 where \( r' = L/R \). (L is the length of the model)

![Figure 11-Z force coefficient with respect to r' for Autosub RA Tests [5]](image)

![Figure 12- Pitching Coefficient with respect to r' for Autosub RA Tests [5]](image)
As can be seen from graph above, experimental data and CFD results have good agreement. Dynamic coefficients are calculated by using below presented formulas.

\[
Z_q' = \frac{Z}{\frac{1}{2} \rho V L^3} \quad Z' = \frac{Z}{\frac{1}{2} \rho V^2 L^2} \quad Z'_q = \frac{Z'V}{qL} \text{ where } V = qR,
\]

By combining \( Z_q' \) and \( Z' \)

\[
Z_q' = \frac{Z'}{\frac{Z'}{r'} R} = \frac{Z'}{r'}
\]  

(7)

By using above equations, it can be said that \( Z'_q \) is the gradient of the trend line of \( Z' \) vs \( r' \) graph. Same procedure is applied for \( M'_q \) and results of presented in Table 1.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>CFD(x10^3)</th>
<th>Experiment(x10^3)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Z'_q )</td>
<td>-12.02</td>
<td>-12.64</td>
<td>4.9</td>
</tr>
<tr>
<td>( M'_q )</td>
<td>-5.8</td>
<td>-5.35</td>
<td>8.4</td>
</tr>
</tbody>
</table>

Table 1-Calculated Dynamic Coefficients and Experimental Data

CFD results and experimental data are in good agreement and errors are smaller than 10%.

**Planar Motion Mechanism (PMM)**

CFD simulations of PMM test are performed in two different subtitles:

- Pure heave motion
- Pure Pitch motion

**Pure Heave Motion:**

Surface and boundary grids are same as straight-line towing test grids for AUTOSUB geometry. However, in fluid domain, an interface is defined as sphere-shape. Fluid domain and volume grid are shown at Fig. 13.

![Figure 13-Fluid Domain and Volume Grid for Pure Heave Test](image)

Boundary conditions of the outer fluid domain are exactly the same. Sphere has interface boundary condition and to achieve pure heave motion. Pre-defined mesh motion is applied to interface and volume meshes around the AUTOSUB geometry. After that, transient analysis is performed. Pre-defined mesh motion is applied to the sphere domain, by using UDF file with following sinusoidal function:

\[
(2\pi af) \times \cos(2\pi ft)
\]

(8)
Where test conditions for pure heave motion simulation:

- Free Stream Velocity \( \rightarrow 2.69 \text{ [m/s]} \)
- Amplitude \([a] \rightarrow 0.1\text{[m]}\)
- Frequency \([f] \rightarrow 1.5\text{[Hz]}\)
- Ambient Pressure \(\rightarrow 150277\text{[Pa]}\)

Boundary conditions of pure heave motion are shown at Fig. 14.

**Figure 14-Fluid Domain and Defined Boundary Conditions for Pure Heave Motion**

Time dependent moment \((M')\) and force coefficients \((Z')\) are obtained from CFD simulation and results are compared to experimental data. Experimental data are obtained by using following equation of motion [6] with known experimental results of dynamic coefficients [7].

\[
Z'(t) = Z'_w w' + Z'_q q' + Z'_\dot{q} \dot{q}'
\]  
\[
M'(t) = M'_w w' + M'_q q' + M'_\dot{q} \dot{q}'
\]

For pure heave motion, \(q\) and \(\dot{q}\) terms are 0. CFD simulation results and experimental data are shown in Fig. 15 and Fig. 16.

**Figure 15 -Z Force Coefficient w.r.t time for Pure Heave Motion Test [8]**

**Figure 16-Pitch Moment Coefficient w.r.t time for Pure Heave Motion Test [8]**

CFD results of dynamic coefficients \((Z'_w, M'_w, Z'_q, \text{ and } M'_q)\) are obtained by regression analyses by using MINITAB commercial program and results are tabulated in Table 2.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>CFD(x10^3)</th>
<th>Experiment(x10^3)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Z'_w)</td>
<td>-33.81</td>
<td>-29.13</td>
<td>16</td>
</tr>
<tr>
<td>(M'_w)</td>
<td>5.57</td>
<td>4.68</td>
<td>25.5</td>
</tr>
<tr>
<td>(Z'_q)</td>
<td>-19.29</td>
<td>-17.39</td>
<td>10.9</td>
</tr>
<tr>
<td>(M'_q)</td>
<td>-0.18</td>
<td>-0.17</td>
<td>4.67</td>
</tr>
</tbody>
</table>

**Table 2-Calculated Dynamic Coefficients from Pure Heave Simulation and Exp. Data**
As shown in Table 2, maximum error is 25.5%. Difference between experimental and numerical results may due to the complexity of the fluid and modelling approach. In advance, dynamic motion mesh and volume definition have to be investigated.

**Pure Pitch Motion:**

For the pure pitch motion analyses, grids and boundary conditions are same as pure heave motion simulation except for interface. In this simulation, two different interfaces are defined and inner domain has mesh motion defined by using an UDF file. In Fig. 17, boundary conditions are shown.

![Figure 17-Fluid Domain and Defined Boundary Conditions for Pure Pitch Motion](image)

Same sinusoidal functions (8) are applied and cruise condition for pure pitch motion simulation:

- Free-stream velocity $\rightarrow$ 2.69 [m/s]
- Amplitude $\rightarrow$ 10 [$^\circ$]
- Frequency $\rightarrow$ 1.5 [Hz]
- Ambient Pressure $\rightarrow$ 150277 [Pa]

CFD results of time dependent force ($Z'$) and moment coefficient ($M'$) are shown at Fig. 18 and Fig. 19 and dynamic coefficients of pure pitch motion are tabulated in Table 3:

![Figure 18-Z Force Coefficient w.r.t time for Pure Pitch Motion Test [8]](image)  ![Figure 19- Pitch Moment Coefficient w.r.t time for Pure Pitch Motion Test [8]](image)

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>CFD($x10^3$)</th>
<th>Experiment($x10^3$)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Z'_q$</td>
<td>-23.6</td>
<td>-11.22</td>
<td>110.0</td>
</tr>
<tr>
<td>$M'_q$</td>
<td>-5.34</td>
<td>-5.04</td>
<td>5.9</td>
</tr>
<tr>
<td>$Z''_q$</td>
<td>-0.293</td>
<td>-0.169</td>
<td>73.4</td>
</tr>
<tr>
<td>$M''_q$</td>
<td>-1.1</td>
<td>-0.98</td>
<td>12.2</td>
</tr>
</tbody>
</table>

**Table 3- Dynamic Coefficients Results of Pure Pitch Motion Simulation and Exp. Data**
As shown in the Table 3, dynamic coefficients cannot be predicted well for this type of analysis but results obtained from Rotating Arm analysis can be used to find dynamic coefficients. Also, CFD results of dynamic moment coefficients and experiments are in good agreement.

CONCLUSION

In this study, captive model test mechanisms such as Straight Towing Tank, RA and PMM have been simulated by CFD techniques using commercial tools. Although the Captive Model Test Mechanisms are the most reliable methods to calculate hydrodynamic characteristics and validate the hydrodynamic design of an underwater vehicle, these methods are the most expensive ones to determine the hydrodynamic coefficients. However, increasing computer power and developed solution methods, CFD analysis are used in the design of AUVs. In this context, a new numerical calculation approach based on CFD is applied to calculate hydrodynamics coefficients of underwater vehicles in 5DOF. Verification study for roll motion is not performed and it is planned as future work. To verify the applied approach, Autosub AUV is used which was performed in the National Oceanography Centre, Southampton. CFD analyses are performed at different cruise and maneuvering conditions. Rotational, sinusoidal and straight line motions of Autosub are simulated. Some rotational terms could also be calculated by PMM analysis but, rotational terms which are calculated by RA simulation are closer to test data.

In conclusion, in the initial design step of hydrodynamic design of an AUV, applied approach can be used to predict hydrodynamic coefficients and hence hydrodynamic characteristic of AUV. However, in the detail design phase, underwater test mechanisms are still have to be used to finalize the hydrodynamic design and hydrodynamic coefficient database.

REFERENCES

1. Data Sheet, Overview of Model Testing Facilities, NTNU.