VNUHCM Journal of Engineering and Technology 2026, 9(3):3107-3115

Open Access Full Text Article

Research Article

Study on the stable dynamics of AUV using computational fluid dynamics (CFD)

Thien-Phuong Ton[1,2,*], Thien-Phuc Tran[1,2], Manh-Dien Huynh[1,2]

[1]*Faculty of Mechanical Engineering, Ho Chi Minh City University of Technology (HCMUT), 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Vietnam*

[2]*Vietnam National University HCMC, Linh Trung Ward, Thu Duc City, Ho Chi Minh City, Vietnam*

Correspondence

Thien-Phuong Ton, Faculty of Mechanical Engineering, Ho Chi Minh City University of Technology (HCMUT), 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Vietnam

Vietnam National University HCMC, Linh Trung Ward, Thu Duc City, Ho Chi Minh City, Vietnam

Email: tonphuong@hcmut.edu.vn

History

- Received: 06-08-2023
- Revised: 09-11-2024
- Accepted: 16-07-2026
- Published Online: 05-08-2026

DOI : https://doi.org/10.32508/vnuhcmj-et.v9i3.1146

Check for updates

ABSTRACT

Understanding the hydrodynamic behavior of Autonomous Underwater Vehicles (AUVs) is essential for evaluating their stability and control. In this study, a computational approach is employed to investigate the stable motion characteristics of an AUV operating at a design speed of 1.5 m/s. The simulation setup replicates procedures similar to those used in the towing tank and rotating arm tests, as standardized by the International Towing Tank Conference (ITTC). The goal is to determine the hydrodynamic derivatives associated with the AUV's motion in both the horizontal and vertical planes. By applying Computational Fluid Dynamics (CFD), the study calculates reaction forces and moments acting on the AUV's body under controlled flow conditions. These values are then used to identify the linear coefficients in the vehicle's equations of motion. Stability analysis is performed using the Routh-Hurwitz criterion to assess straight-line stability at the selected operating speed. The CFD simulations are performed using ANSYS FLUENT, a widely used and robust commercial software for fluid dynamics problems. The fluid domain is modeled under steady conditions, and the turbulence effects are accounted for using the Reynolds-Averaged Navier–Stokes (RANS) model. Near-wall effects, which are critical for accurately predicting pressure distributions and shear forces, are handled using the standard k–ε turbulence model. This combination offers a practical compromise between computational time and accuracy, especially when assessing pressure and shear distributions around the hull. The study demonstrates that CFD can be effectively used to evaluate dynamic stability in the early stages of AUV design. This method reduces the need for costly physical testing and provides designers with critical insights into hydrodynamic behavior, which is beneficial for optimizing control systems and improving vehicle performance in underwater missions.

Key words: straight-line stability, Routh criteria, AUV, CFD

INTRODUCTION

Kinematic coefficients such as damping force and added-mass factors are physical properties associated with the profile of the AUV and are expressed in the dynamic equations of the AUV. Some experimental methods to determine these coefficients include the steady rotating arm experiment (the subject is mounted on a rotating arm), AUV oscillation in the vertical or horizontal plane, and PMM [1]. However, those experimental methods require spacious facilities and expensive equipment. A CFD simulation to determine the kinetic coefficients has been performed by many authors to investigate the interaction between the AUV and the fluid flow. The CFD simulation model often reproduces the methods specified in the experimental procedures guide issued by the ITTC (2005) [2]. There have been many academic papers about simulating AUV rotation around one fixed point with a given radius, ignoring the effect of external flow. For instance, Zhang et al. simulate the flow when rotating an AUV profile with a constant radius regardless of time to determine the total impact force on the AUV body while changing its sliding angle as well as to inspect the vortex adjacent to the body [3]. Racine and Paterson simulate a NNEMO diving device rotating around a fixed point, taking into account the effect of time, using the overset-mesh method for mesh generation to determine the hydrodynamic coefficients of AUV NEMO [4]. This method requires a lot of computational resources and can be considered time-consuming to determine the mesh size at the transition regions. The AUV design consists of two wings on both sides for the self-propelled mode, which takes advantage of the gravitational effect and reduces the time the AUV returns to the water surface. This research focuses on the sum of the impact forces on the AUV using the CFD method. The simulations are created and solved using the commercial software ANSYS Fluent. Furthermore, this research also focuses on solving for the kinematic coefficients in the system of equations describing the motion of the AUV; these coefficients will help evaluate the im-

Cite this article : Ton T, Tran T, Huynh M. **Study on the stable dynamics of AUV using computational fluid dynamics (CFD)**. *VNUHCM J. Eng. Technol.* 2026; 9(3):3107-3115.

pact of the side wings on the stability of the AUV—the ability to maintain the advancing direction of the AUV under disturbances in the yaw direction. The AUV 2000 design consists of two wings on both sides for the self-propelled mode, which takes advantage of the gravitational effect and reduces the time the AUV returns to the water surface. This research focuses on the sum of the impact forces on the AUV using the CFD method. The simulations are created and solved using the commercial software ANSYS Fluent. Furthermore, this research also focuses on solving for the kinematic coefficients in the system of equations describing the motion of the AUV; these coefficients will help evaluate the impact of the side wings on the stability of the AUV—the ability to maintain the advancing direction of the AUV under disturbances in the Yaw direction. Therefore, the authors can evaluate the maneuverability and control ability of the AUV 2000 when there is no external flow and no rudder steering.

METHODS OF CFD SIMULATION MODELING AND PROCESSING OF SIMULATION RESULTS

The geometry of AUV

The AUV is designed in accordance with the Myring shape[5], comprising three primary sections: the head section, the body section, and the tail section. Detailed representations of its form and dimensions are provided in Figure 1 and Table 1 below.

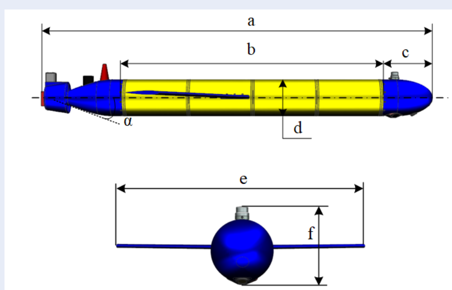

Figure 1: Displaying the projections and dimensions of the AUV

where n are parameters to control the curve shape of the AUV's head, with the AUV's head shape described by equations such as:

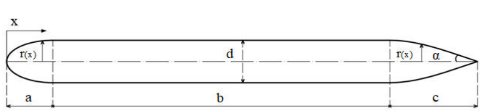

Table 1: Basic parameters of the AUV

Parameter	Value	Unit
a	2650	mm
b	1790	mm
c	245	mm
d	250	mm
α	20	degrees
e	1450	mm
f	320	mm
n	2	

where :

$$r(x) = \frac{1}{2} d \left[1 - \left(\frac{x-a}{a} \right)^2 \right]^{\frac{1}{n}}$$

For the purpose of facilitating simulation presentation, we establish a coordinate system to assess the maneuverability of the AUV, as shown in Figure 2.

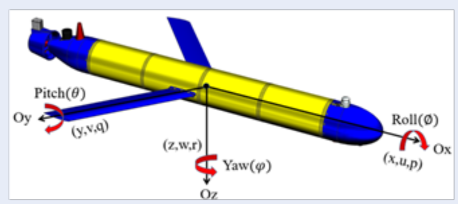

Figure 2: The coordinate system used to examine the maneuverability of the AUV

Linear stability when moving in the horizontal plane

After omitting the non-considered degrees of freedom from the ordinary equation describing the motion with six degrees of freedom (Fossen, 2011, p. 8)[6], the new system of equations only describe the motion of the AUV in the horizontal plane includes surge (transitional motion in the xaxis), sway (translational motion in the y axis), yaw (rotational motion around the z axis), a coordinate system rigidly attached to the AUV's body and is positioned exactly at its center of gravity. (Fossen, 2011, p. 9) .

$$\begin{cases} m(\dot{u} - vr) = X_{CFD} \\ m(\dot{u} - vr) = Y_{CFD} \\ I_{zz}\dot{r} = N_{CFD} \end{cases} \quad (01)$$

ince the operating speed of the AUV is only 1 to 3 knots, the external impact force on the AUV in the horizontal plane can be linearized through the Taylor expansion as follows (Lewis, 1988, pp. 217-233)

$$\begin{cases} Y = Y_{CDF} = v\frac{\partial Y}{\partial v} + \dot{v}\frac{\partial Y}{\partial \dot{v}} + r\frac{\partial Y}{\partial r} + \dot{r}\frac{\partial Y}{\partial \dot{r}} \\ N = N_{CFD} = v\frac{\partial N}{\partial v} + \dot{v}\frac{\partial N}{\partial \dot{v}} + r\frac{\partial N}{\partial r} + \dot{r}\frac{\partial N}{\partial \dot{r}} \end{cases} \quad (02)$$

If only sway and yaw motions are considered, then by applying the Routh stability criterion, we can obtained the following conditions for the AUV to achieve stability (Fossen, 2011, p. 351; Lewis, 1988, pp. 199-201)

$$N_r Y_v - N_v (Y_r - mU) > 0 \quad (03)$$

$$\Leftrightarrow \frac{N_r}{Y_r - mU} - \frac{N_v}{Y_v} > 0 \quad (04)$$

Note:
m is the total mass of AUV
U is the magnitude of the total velocity vector
Y_v is the partial derivative of Y with respect to $v\left(\frac{\partial Y}{\partial v}\right)$
Y_r is the partial derivative of Y with respect to $r\left(\frac{\partial Y}{\partial r}\right)$
N_v is the partial derivative of N with respect to $v\left(\frac{\partial N}{\partial v}\right)$
N_r is the partial derivative of N with respect to $r\left(\frac{\partial N}{\partial r}\right)$
These results can be obtain using the following nondimensionalization formula:

$$m' = \frac{m}{\left(\frac{1}{2}\right)\rho L^3},\ v' = \frac{v}{u},\ r' = \frac{rL}{u},\ Y_v' = \frac{Y_v}{\left(\frac{1}{2}\right)\rho L^2 u},$$
$$N_v' = \frac{N_v}{\left(\frac{1}{2}\right)\rho L^3 U},\ Y_r' = \frac{Y_v}{\left(\frac{1}{2}\right)\rho L^3 U}\ and\ N_r' = \frac{N_v}{\left(\frac{1}{2}\right)\rho L^4 U}$$

By applying Routh stability criterion, thefollowing in-Equation (05) can be obtained:

$$\Leftrightarrow \frac{N_r'}{Y_r' - m'} - \frac{N_v'}{Y_v'} > 0 \quad (05)$$

Rotating stability when moving in the horizontal plane

The investigated AUV's motions include surge (translational motion in the x_b axis), sway (translational motion in the y_b axis), yaw rotation around z_b. (Minnick, 2006, p. 20)[7].

$$\begin{cases} m(\dot{w} - qU) = Z_{CFD} \\ I_{xx}\dot{P} = K_{CFD} \\ I_{yy}\dot{q} = M_{CFD} \end{cases} \quad (06)$$

Because of the AUV low speed, a linear model can be used to describe the AUV moving in the vertical plane

$$\begin{cases} Z = Z_{CFD} = w\frac{\partial Z}{\partial w} + \dot{w}\frac{\partial Z}{\partial \dot{w}} + q\frac{\partial Z}{\partial q} + \dot{q}\frac{\partial Z}{\partial \dot{q}} \\ M = M_{CFD} = w\frac{\partial M}{\partial w} + \dot{w}\frac{\partial M}{\partial \dot{w}} + q\frac{\partial M}{\partial q} + \dot{q}\frac{\partial M}{\partial \dot{q}} \end{cases} \quad (07)$$

By ignoring the roll motion, then adding Equation (06) to Equation (07) and apply the Routh stability criterion, the stability condition for the AUV movement in the vertical plane can be obtained (Minnick, 2006):

$$\frac{m_q}{Z_q + mU} - \frac{M_w}{Z_w} > 0 \quad (08)$$

Where:
Z_q is the partial derivative of Z with respect to $q\left(\frac{\partial z}{\partial q}\right)$
M_q is the partial derivative of M with respect to $q\left(\frac{\partial M}{\partial q}\right)$
Z_w is the partial derivative of Z with respect to $w\left(\frac{\partial Z}{\partial w}\right)$
M_w is the partial derivative of M with respect to $w\left(\frac{\partial M}{\partial w}\right)$
These results can be obtain using the following nondimensionalization formula:

$$m' = \frac{m}{\left(\frac{1}{2}\right)\rho L^3},\ w' = \frac{w}{U},\ q' = \frac{qL}{U},\ Z_w' = \frac{Z_w}{\left(\frac{1}{2}\right)\rho L^2 U},$$
$$M_w' = \frac{M_w}{\left(\frac{1}{2}\right)\rho L^3 U},\ Z_q' = \frac{Z_q}{\left(\frac{1}{2}\right)\rho L^3 U}\ and\ M_q' = \frac{M_w}{\left(\frac{1}{2}\right)\rho L^4 U}$$

Công thức
By applying Routh stability criterion, the following in-Equation (09) can be obtained:

$$\frac{M'q}{Z'q + m} - \frac{M'w}{Z'w} > 0 \quad (09)$$

CFD SIMULATION PROPERTIES AND MESH GENERATION

ANSYS Fluent is used to calculate the fluid flow around the AUV. The AUV is put under a rotating reference frame (Single Rotating Reference Frames (SRF)), and the speed, pressure, etc. variables are discrete in the second-order upwind differencing scheme. The least-squares cell-based method is used to determine the gradients. The RANS model is used because it is computationally simple, generates the results in a time-effective manner, is suitable for the research aims of this paper, and has an acceptable accuracy for industrial applications (Argyropoulos & Markatos, 2015)[8]. The k-epsilon turbulent model is used to model turbulent flow because it is suitable for modeling flows subject to high shear stress near the AUV body profile, moderate energy eddy flows, and local transitional flows (stratum boundary layer, etc.) (Song et al., 2018)[9]. After setting up the computational domain, we imported the complete 3D model of the AUV into the Fluent environment of Ansys 2020R1 for meshing. For each distinct state of the AUV, such as changes in pitch and yaw angles during straight-line movement or changes in pitch and yaw angles during turns, a separate mesh model was created. These mesh models were generated using polyhedral meshing; however, specifically, hexahedral meshing was utilized due to its ability to provide good element quality and reduce the number of nodes, thereby reducing simulation time. The boundary layer mesh was divided into a prism mesh comprising five layers, with the first layer having a thickness of 1mm and a growth rate of 1.2. The mesh was refined near the AUV's surface and coarsened towards the outer shell. Mesh quality was assessed based on criteria such as Non-Orthogonality and Equiangle Skewness. The meshing results are displayed in Figure 3.

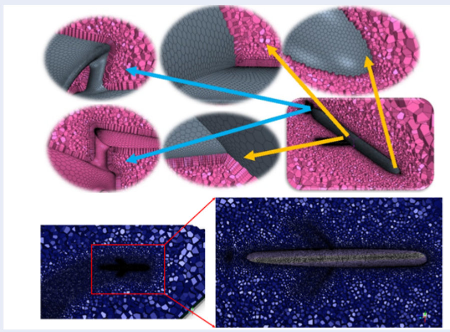

Figure 3: Meshgeneration result

Boundary condition properties

The shape of the flow region is rectangular when it is necessary to calculate the coefficients representing the changes of impact force and torque in accordance with translational velocity in the direction of velocities v and w. The size and condition of the boundary are shown in Figure 4.

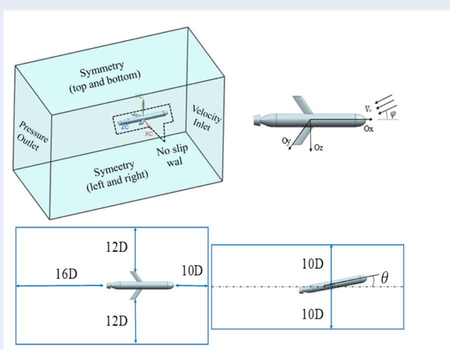

Figure 4: Flow region for AUV translational motion

Where it is necessary to determine the coefficient representing the change of impact force and moment with respect to angular velocities, the shape of the flow region is the disk sector in the case of R < 2.5L (L is the length of the AUV) and is an annulus if R > 2.5L. The center of the inertial coordinate system is set to coincide with the geometric center of the AUV body, and the center of rotation is set at a distance R from the origin of the inertia. To solve for turning radius R and angular velocity in simulation, we choose the value of dimensionless angular velocity in the range 0.2 to 0.5 (Feldman, 1987)[10], and the tangential velocity U is set at 1.5 knots (as the designed speed of AUV) (Sen, 2011)[11]. From these selected values, we can determine the turning radius R used in the simulation.

The size and condition of the boundary are shown in Figure 5. The zero-constant-velocity boundary condition is established for the Velocity inlet. The pressure outlet, which is the outlet of fluid flow, is set at an absolute static pressure of 5 bar. The remaining faces have boundary conditions for symmetry.

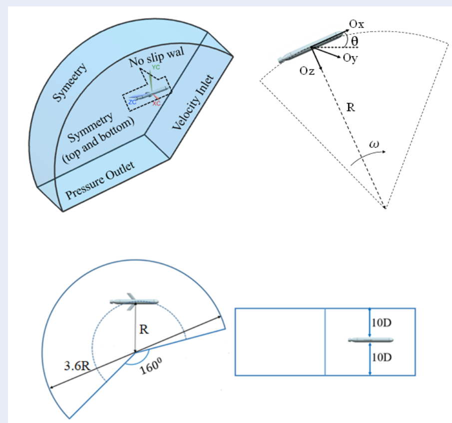

Figure 5: Flow region for AUV rotatingmotion

RESULTS

Result for AUV motion in the horizontal plane

The simulation results in the horizontal plane are illustrated through Figure 6 to Figure 13, covering both translational and rotational motion scenarios of the AUV. These results are fundamental for estimating hydrodynamic derivatives and analyzing the dynamic response of the vehicle. Figure 6 and Figure 7 present the variation of lateral force Y' and yawing moment N' as functions of the non-dimensional forward velocity v'. These relationships are essential for determining sway force and yaw moment coefficients associated with translational motion. Complementary to these, Figure 8 and Figure 9 visualize the velocity and pressure contours when the AUV is moving in a straight line with a 10-degree yaw deviation. The asymmetric flow separation and pressure fields observed in these figures illustrate the source of hydrodynamic forces and moments acting on the vehicle during off-axis motion. Figure 10 and Figure 11 show the computed lateral force Y′ and yawing moment N′ in response to the angular velocity r′, representing the hydrodynamic behavior during rotational motion about a fixed center. These datasets are critical for evaluating rotational damping and added moment coefficients in the yaw direction. Supporting these numerical data, Figure 12 and Figure 13 provide flow field visualizations in terms of velocity and pressure contours under rotational motion. The flow asymmetries and high-pressure zones near the hull confirm the trends observed in force and moment outputs.

The coefficients Y_v and N_v are obtained using regression on statistics representing the change of Y and N with respect to velocity v (v=U sinφ) while the AUV is advancing at U = knots with shear angles φ from 0 to 10 degrees in 2.5-degree increments. Then, we nondimensionalize Y_v and N_v to obtain Y_v' and N_v' using the following formulas:

$$Y_v' \frac{Y_v}{\frac{1}{2}pL^2U'}, \; N_v' = \frac{N_v}{\frac{1}{2}pL^3U}$$

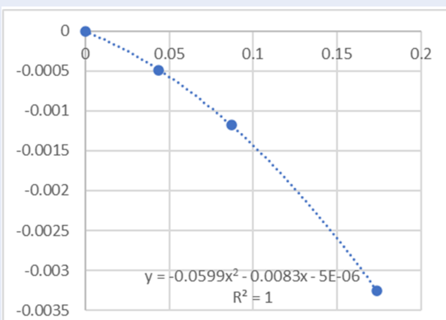

Figure 6: Force Y′ as a Function of velocity v′

The coefficients Y_r and N_r are obtained using regression on statistics representing the change of Y and N with respect to angular velocity while the AUV is advancing at U = 3 knots with shear angles β equals 0. Then, we nondimensionalize Y_r and N_r to obtain Y_r' and N_r' using the following formulas:

$$Y_r' \frac{Y_r}{\frac{1}{2}pL^3U'}, \; N_r' = \frac{N_r}{\frac{1}{2}pL^4U}$$

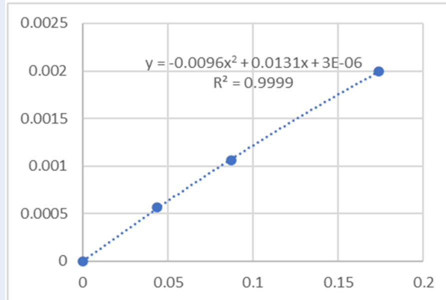

Figure 7: Torque N′ as a Function of velocity v′

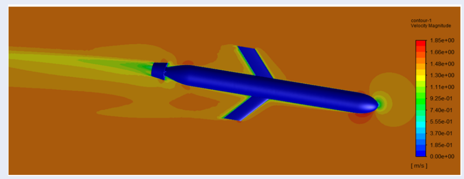

Figure 8: Velocity contour of the fluid when the AUV is intranslational motion with an angle of deviation of 10 degrees in the horizontal plane

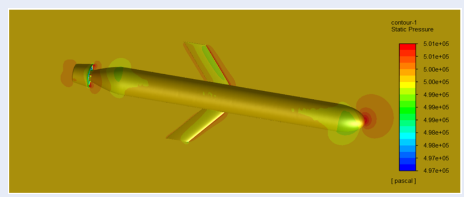

Figure 9: Pressurecontour of the fluid as the AUV rotates translationally with an angle of 10 degrees in the horizontal plane

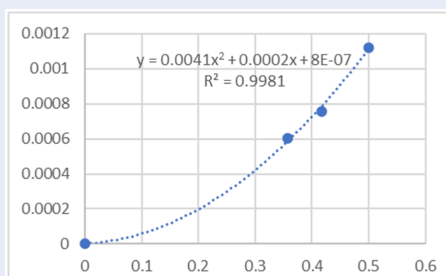

Figure 10: Force Y′ as a Function of angular velocity r′

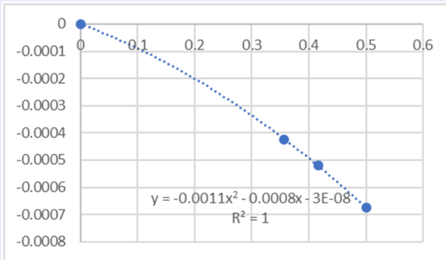

Figure 11: Torque N′ as a Function of angularvelocity r′

Table 2: **Linearhydrodynamic coefficients of AUV motion in the horizontal plane**

Hydrodynamic coefficients	Value
Y_v'	-0.0083
N_v'	0.013
Y_r'	0.0002
N_r'	- 0.0008

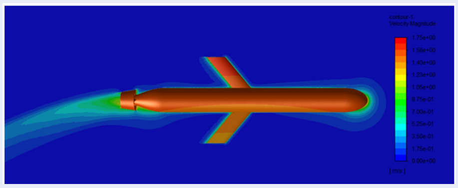

Figure 12: Velocity contour of the fluid as the AUV rotates around afixed center in the horizontal plane

Figure 13: Pressure contour of the fluid as the AUV rotates around afixed center in the horizontal plane

The hydrodynamic coefficients in Table 2 are added to Equation (05) to determine the stability of the AUV motion in the horizontal plane. These hydrodynamic coefficients are obtained through the use of a first-order Taylor series expansion in regression analysis of the forces and moments acting on the AUV. These functions are represented in Figures 6, 7 and 10 and 11.

Result for AUV motion in the vertical plane

The vertical plane simulation results, presented in Figure 14 through Figure 21, provide key insights into the hydrodynamic behavior of the AUV under both translational and rotational motions in the pitch direction. These simulations are conducted to extract hydrodynamic derivatives associated with heave and pitch dynamics, which are vital for assessing vertical plane stability and control performance.

Figure 14 and Figure 15 show the variation of vertical force Z′ and pitching moment M′ as functions of the vertical velocity w′, corresponding to heave motion with different angles of attack. These force and moment responses are used to estimate added mass and damping coefficients in the vertical direction. To support the force and moment analysis, Figure 16 and Figure 17 illustrate the velocity and pressure contour fields when the AUV is in translational motion with a varying angle of attack. The observed changes in flow separation, wake region, and pressure gradients indicate how the fluid field reacts to pitch-induced body inclination. Rotational motion analysis is detailed in Figure 18 and Figure 19, where the relationships between Z′ and pitching angular velocity q′, as well as M′ and q′, are plotted. These results allow for the identification of pitch-related hydrodynamic coefficients that govern the vehicle's response to rotational excitation. Figure 20 and Figure 21 complement these findings by providing velocity and pressure contours as the AUV rotates about a fixed transverse axis in the vertical plane. The symmetry-breaking flow structures and localized pressure zones confirm the mechanisms generating the observed moments and forces. Similar to the AUV motion in the horizontal plane, the coefficients Z_w and M_w are obtained using regression on statistics representing the change of Z and M with respect to velocity w (w= U sinθ) while the AUV is advancing at U = 3 knots with pitch angles $\theta$from 0 to 10 degrees in 2.5-degree increments. Then, we nondimensionalize Z_w and M_w to obtain Z_w' and M_w' using the following formulas:

$$Z_w' \frac{Z_v}{\frac{1}{2}pL^2U'}, \; M_w' = \frac{M_w}{\frac{1}{2}pL^3U}$$

The coefficients Z_q and M_q are obtained using regression on statistics representing the change of Y and N with respect to angular velocity r while the AUV is advancing at U = 3 knots with pitch angles θ equals 0.

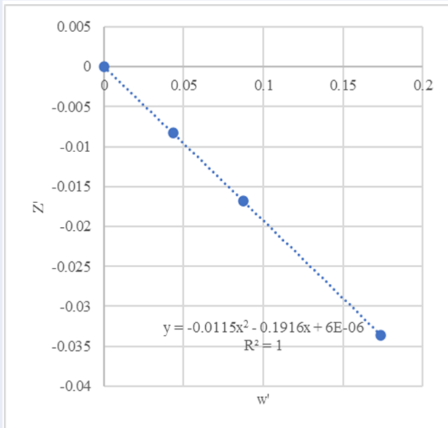

Figure 14: Force Z′ as a Function of velocity w′

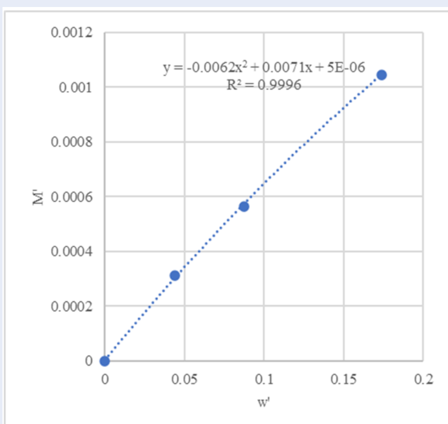

Figure 15: Torque M′ as a Function of velocity w′

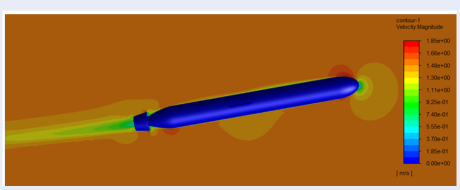

Figure 16: Velocity contour of the fluid when the AUV is intranslational motion with a variable angle of attack

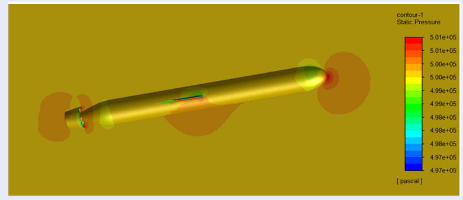

Figure 17: Pressure contour of the fluid when the AUV is intranslational motion with a variable angle of attack

Then, we nondimensionalize Z_q and M_q to obtain $Z_q^{'}$ and $M_q^{'}$ using the following formulas:

$$Z_q^{'}\frac{Z_q}{\frac{1}{2}pL^3U^{'}},\; M_q^{'}=\frac{M_q}{\frac{1}{2}pL^4U}$$

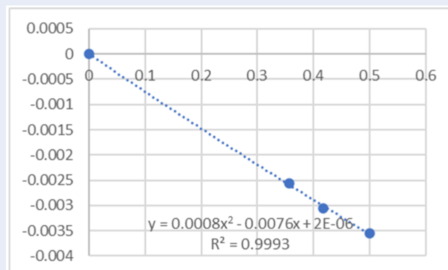

Figure 18: Force Z′ as a Function of angularvelocity q′

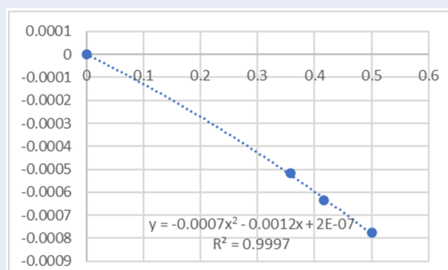

Figure 19: Torque M′ as a Function of angular velocity q′

The hydrodynamic coefficients in Table 3 are added to Equation (09) to determine the stability of the AUV in the vertical plane.

DISCUSSION AND CONCLUSION

CFD simulations and Routh criteria are used to investigate the stability of the AUV in the vertical and horizontal planes during motion without external flow

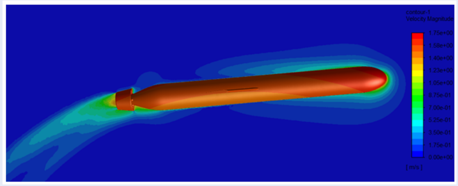

Figure 20: Velocity contour of the fluid as the AUV rotates around afixed center in the vertical plane

Figure 21: Pressure contour of the fluid as the AUV rotates around afixed center in the vertical plane

Table 3: Linear hydrodynamic coefficients of AUV motion in the vertical plane

Hydrodynamic coefficients	Value
Z'_w	-0.19
M'_w	0.0071
Y'_v	0.0076
N'_v	0.0012

and how impact force affect the original orbit of the AUV in the yaw direction and itch when there is no steering control using the rudder.

To determine the change of force and torque with respect to yaw velocity, the AUV is simulated in advancing motion with various shear angles and deflections. When simulating an advancing AUV, we need six independent simulations instead of 18 while varying the shear angle and deflection to determine the kinematic coefficients in both the vertical and horizontal planes (ITTC, 2005)[2]. By varying the radius and tangential velocity while keeping the angular speed constant, we obtain force and torque values that vary with tangential velocity. The number of simulations is 12 to meet the original requirements. Then, the hydrodynamic coefficients (viscosity coefficient) obtained from Table 2 and Table 3 are determined through the use of a first-order Taylor series expansion in regression analysis of the forces and moments acting on the AUV.

Simulation results show that the AUV is stable when moving in the horizontal plane but unstable in the vertical plane when no steering controls are involved. To increase the stability of the AUV, we can add additional glider wings to the rear of the AUV. The glider wings would create drag that stabilizes the AUV's movement. They would function as fletching on arrows.

ACKNOWLEDGEMENT

We acknowledge Ho Chi Minh City University of Technology (HCMUT), VNU-HCM for supporting this study.

CONFLICT OF INTEREST

The authors declare that they have no conflicts of interest regarding the publication of this article.

AUTHOR CONTRIBUTIONS

Ton Thien Phuong conceived the study and drafted the manuscript.

Huynh Manh Dien conducted the simulations presented in the study and contributed to writing the manuscript.

Tran Thien Phuc contributed to revising and editing the manuscript.

REFERENCES

1. Edwward V, Lewis. Principles of Naval Architecture -Volume II - Resistance, Propulsion and Vibration. 1988;.
2. ITTC. Testing and Extrapolation Methods Manoeuvrability Captive Model Test Procedures. 2005;.
3. Zhang JT, Maxwell JA, Gerber AG, Holloway AGL, Watt GD. Simulation of the flow over axisymmetric submarine hulls in steady turning. Ocean Engineering. 2013;57:180–196.
4. Racine B, Paterson E. CFD-based method for simulation of marine-vehicle maneuvering. 35th AIAA Fluid Dynamics Conference and Exhibit. 2005;.
5. Myring DF. A Theoretical Study of Body Drag in Subcritical Axisymmetric Flow. 1976;.
6. Fossen TI. Handbook of Marine Craft Hydrodynamics and Motion Control. New York: Wiley; 2011.
7. Minnick L. A parametric model for predicting submarine dynamic stability in early stage design Virginia Tech. 2006;.
8. Argyropoulos CD, Markatos NC. Recent advances on the numerical modelling of turbulent flows. Applied Mathematical Modelling. 2015;39(2):693–732.
9. Song J, Kharoua N, Khezzar L, Alshehhi M. RANS Modelling of a Swirling Flow Interacting With a Conical Bluff Body ASME 2018 5th Joint US-European Fluids Engineering Division Summer. 2018;.
10. Feldman J. Straightline and rotating arm captive-model experiments to investigate the stability and control characteristics of submarines and other submerged vehicles. 1987;.
11. Sen D. Mod-01 Lec-34 Capative Model Tests and Experimental Determination of Hydrodynamic Derivatives. NPTEL. 2011;.

VNUHCM Journal of Engineering and Technology 2026, 9(3):3107-3115

Open Access Full Text Article

Bài nghiên cứu

Nghiên cứu sự ổn định động học của tàu lặn tự hành (AUV) bằng mô phỏng động lực học chất lỏng (CFD)

Tôn Thiện Phương[1,2,*], Trần Thiên Phúc[1,2], Huỳnh Mạnh Diễn[1,2]

TÓM TẮT

Nghiên cứu đặc tính thủy động học của phương tiện lặn tự hành (AUV) đóng vai trò then chốt trong việc đánh giá khả năng ổn định và điều khiển của thiết bị khi hoạt động dưới nước. Trong nghiên cứu này, phương pháp mô phỏng số được sử dụng để phân tích chuyển động ổn định của AUV ở tốc độ thiết kế 1,5 m/s. Quy trình mô phỏng được xây dựng dựa trên nguyên lý của các thử nghiệm thực nghiệm theo khuyến nghị của ITTC, nhằm đảm bảo độ tin cậy trong việc xác định các hệ số trong mô hình liên quan đến chuyển động của AUV trong hai mặt phẳng ngang và dọc. Thông qua mô phỏng CFD, lực và mô men tác dụng lên thân AUV được tính toán trong điều kiện dòng chảy được kiểm soát. Từ các kết quả này, các hệ số tuyến tính trong mô hình phương trình chuyển động được tính toán. Đồng thời, tiêu chuẩn Routh-Hurwitz được áp dụng để kiểm tra ổn định chuyển động thẳng của AUV tại vận tốc khảo sát. Quá trình mô phỏng được thực hiện trên phần mềm ANSYS FLUENT. Miền dòng chảy được giả lập ở trạng thái ổn định, và mô hình RANS được sử dụng để xét đến hiệu ứng của nhiễu động. Ảnh hưởng gần thành, rất quan trọng trong việc dự đoán phân bố áp suất và lực cắt, được mô hình hóa bằng mô hình k–ε tiêu chuẩn nhằm nâng cao độ chính xác trong việc mô phỏng phân bố áp suất và ứng suất cắt dọc theo bề mặt thân tàu. Kết quả nghiên cứu cho thấy phương pháp CFD có thể áp dụng hiệu quả trong giai đoạn thiết kế ban đầu, giúp giảm thiểu sự phụ thuộc vào thử nghiệm thực tế, đồng thời cung cấp cơ sở dữ liệu tin cậy cho việc phân tích ổn định và từ đó hỗ trợ tối ưu hóa hệ thống điều khiển và nâng cao hiệu suất hoạt động của phương tiện lặn trong môi trường dưới nước.

Từ khoá: Tính ổn định đi thẳng, Tiêu chuẩn ổn định Routh, Phương tiện tự hành dưới nước (AUV), Mô phỏng động học lưu chất (CFD)

[1]*Khoa Cơ Khí, Đại Học Bách Khoa TP.HCM, 268 Lý Thường Kiệt, Quận 10, TP.HCM, Việt Nam*

[2]*Trường Đại học Quốc gia Thành phố Hồ Chí Minh*

Liên hệ

Tôn Thiện Phương, Khoa Cơ Khí, Đại Học Bách Khoa TP.HCM, 268 Lý Thường Kiệt, Quận 10, TP.HCM, Việt Nam

Trường Đại học Quốc gia Thành phố Hồ Chí Minh

Email: tonphuong@hcmut.edu.vn

Lịch sử
- Ngày nhận: 06-08-2023
- Ngày sửa đổi: 09-11-2024
- Ngày chấp nhận: 16-07-2026
- Ngày đăng: 05-08-2026

DOI : https://doi.org/10.32508/vnuhcmj-et.v9i3.1146

Check for updates

Trích dẫn bài báo này: Phương T T, Phúc T T, Diễn H M. **Nghiên cứu sự ổn định động học của tàu lặn tự hành (AUV) bằng mô phỏng động lực học chất lỏng (CFD)** . *VNUHCM J. Eng. Technol.* 2026; 9(3): 3107-3115.