

CFD Analysis of Flow Characteristics in a Rectangular Fluidic Thrust Vectoring Nozzle for Underwater Vehicles

Yiran Chen

*College of Aerospace Engineering, Chongqing University, Chongqing, China
13015553310@163.com*

Abstract. Underwater vehicles are widely applied in marine resource exploitation and marine scientific investigation, yet maneuvering performance and control accuracy remain two prominent technical bottlenecks restricting their operational capability. Fluidic thrust vector control technology relies on the Coanda effect to make primary jet flow cling to curved wall surfaces for flow deflection, which completely removes the requirement for mechanical rotating components. This technical scheme features fast response speed, stable vector control performance under full velocity operating conditions and compact overall layout, making it a promising substitute for traditional mechanical control structures. In this paper, a three-dimensional computational fluid dynamics (CFD) calculation model is established targeting a rectangular nozzle with an aspect ratio of 2:1, and numerical solution calculations are carried out. Under steady-state neutral jet working conditions, obvious trailing-edge suction vortex structures can be observed at the Z-direction 50% central cross-section, which occupy a large core flow area inside the flow field. When the main jet adheres to the deflection wall surface under asymmetric secondary flow supply, an obvious flow separation bubble will form on the wall side where the jet attaches.

Keywords: underwater vehicle, fluid-driven thrust vectoring, Coanda effect, CFD numerical simulation, separation bubble

1. Introduction

1.1. Research on underwater vehicle propulsion and maneuvering technologies

Traditional underwater vehicles realize multi-degree-of-freedom attitude and position adjustment by matching main propulsion systems with independent control rudders [1]. This structural form has multiple inherent defects: the transmission system contains numerous mechanical parts with complex structural design; the control response lags significantly under low navigation speed; mechanical friction and vibration during operation generate extra hydrodynamic noise that weakens vehicle concealment performance; besides, rudder devices and their matching driving mechanisms occupy a large amount of internal cabin space.

To solve the above limitations, many scholars have proposed integrated propulsion schemes that combine thrust output and flow deflection control. The mainstream technical routes put forward at present are sorted out as follows:

(1) Pod-type propulsion system

The pod propulsion structure installs an alternating current driving motor inside a streamlined fairing, and the propeller is directly connected to the motor rotor for synchronous rotation [2]. This layout can save internal cabin space of the vehicle and reduce navigation resistance. However, the propeller needs to operate at a high rotating speed, which easily induces cavitation around the blade surface. Meanwhile, residual swirling flow in the wake region will reduce the overall propulsion efficiency of the system.

(2) Flexible transmission shaft active propulsion technology

Different from the rigid transmission shaft adopted by conventional propulsion systems, this scheme replaces it with flexible shafts. The flexible structure can adjust the output direction and magnitude of thrust in real time during vehicle operation. The whole system operates with low noise level, which helps improve acoustic stealth performance and raise the critical cavitation speed threshold of the vehicle.

(3) Pump-jet propulsion system

The pump-jet structure integrates the driving motor and shrouded impeller as a whole, with compact overall size and outstanding performance in noise suppression and cavitation inhibition. This propulsion layout has been widely adopted in torpedo propulsion equipment for decades [3-5].

1.2. Introduction to fluid-based thrust vectoring technology

Different from the mechanical vector scheme that relies on nozzle rotation, fluidic thrust vectoring technology adjusts the flow state of auxiliary control flow to change the deflection direction of the main jet. The propulsion device can simultaneously output thrust and control torque without any mechanical rotating or swinging parts.

The Coanda effect vector control method arranges a curved trailing-edge wall at the nozzle outlet. The main jet automatically attaches to the curved wall surface under pressure difference and completes deflection. This control mode has simple structural composition, convenient flow regulation characteristics and low hydrodynamic radiation noise.

1.3. Application of fluid-based thrust vectoring in underwater vehicles

At present, mechanical gimbal rotating joints are still the most widely adopted thrust vectoring mode in underwater robots, while relevant research on flow-controlled vector propulsion schemes is relatively scarce.

Compared with shock wave vector control structures, the Coanda effect fluidic vector scheme with co-flow or counter-flow auxiliary injection can achieve larger jet deflection angles and possess higher control efficiency. Nevertheless, this technology still has obvious flow instability risks: the main jet may be uncontrollably adsorbed on one side wall, or suddenly separate from the wall surface with abrupt deflection angle changes. With the gradual deepening of relevant research, such unstable flow phenomena have attracted more and more scholars' research attention.

2. Introduction to computational fluid dynamics

2.1. Overview of computational fluid dynamics

CFD discretizes the continuous fluid calculation domain and solves mass, momentum and energy conservation governing equations to obtain the spatial distribution of each flow field physical

parameter inside the calculation area.

2.2. The three fundamental equations of fluid mechanics

2.2.1. Continuity equation

The continuity equation is established based on the mass conservation law, which means the mass accumulation rate within a fixed control volume equals the net mass flow passing through the volume boundary.

The differential form of the continuity equation is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (1)$$

2.2.2. Momentum equation

The momentum equation balances the rate of change of momentum within a control volume against the surface and body forces acting on it. For a Newtonian fluid this relation reduces to the Navier-Stokes equations.

Taking the direction of the x as an example, the differential form of the momentum equation is:

$$\begin{aligned} \frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u u)}{\partial x} + \frac{\partial(\rho u v)}{\partial y} + \frac{\partial(\rho u w)}{\partial z} = & -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} [2\mu \frac{\partial u}{\partial x} + \lambda(\nabla \cdot \vec{V})] + \frac{\partial}{\partial y} [\mu(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})] \\ & + \frac{\partial}{\partial z} [\mu(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x})] + \rho f_x \end{aligned} \quad (2)$$

2.2.3. Energy equation

The energy equation accounts for conservation of total energy: temporal changes in stored energy equal the mechanical work of external forces plus the net heat flux entering the volume.

The differential form of the energy equation is:

$$\frac{\partial}{\partial t} [\rho(e + \frac{V^2}{2})] + \frac{\partial}{\partial x_j} [\rho u_j(e + \frac{V^2}{2})] = \frac{\partial}{\partial x_j} (k \frac{\partial T}{\partial x_j}) + \frac{\partial}{\partial x_j} (u_i \tau_{ij}) + \rho f_j u_j \quad (3)$$

Where e is the internal energy per unit mass of the fluid, k is the thermal conductivity, and T is the temperature.

3. Flow field analysis of rectangular nozzles with different longitudinal cross-sections

This chapter adopts CFD numerical simulation methods to carry out detailed analysis on the internal flow field of passive fluidic thrust vector nozzles for underwater vehicles. The research object is a nozzle with rectangular outlet structure: the inlet height of the main jet is 20 mm, the outlet aspect ratio is 2:1, the length of Coanda curved wall is 40 mm, the wall inclination angle is set as 14°, and the inlet height of external auxiliary flow is 4 mm, and the geometric model is displayed in Figure 1.

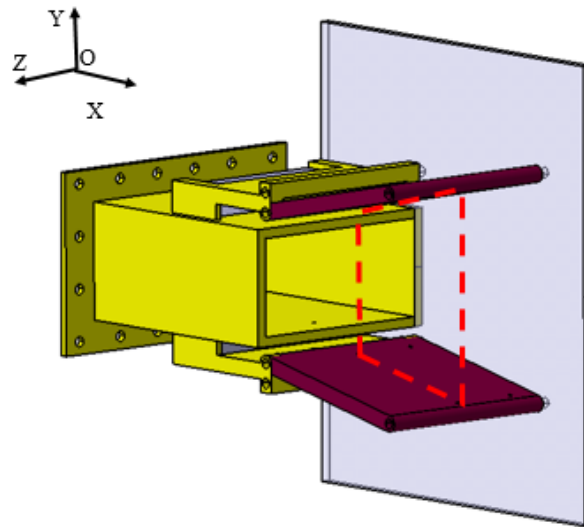


Figure 1. Calculation model

3.1. Steady-state neutral condition of the main jet

When the valves on both sides of the nozzle are open, external fluid flows in from both the top and bottom, and the main jet is in a steady-state neutral condition, producing no deflection. Figure 2 shows the CFD simulation results for the Z-50% cross-section. The velocity contour plot reveals that after the external fluid flows out from the inlet slit, it collides with the Coanda wall to form small-scale vortices. Due to the entrainment effect of the main jet, a portion of the external fluid is drawn into the main jet, where it gradually dissipates downstream until it disappears. At the trailing edge of the Coanda wall, a distinct suction vortex is present. This vortex continuously moves upstream, creating a recirculation phenomenon in the near-wall region. Both the upper and lower walls experience fluid suction at their trailing edges, supplemented by inflow from the external fluid. As a result, the pressures on the upper and lower walls are consistent, and the main jet remains in a steady-state neutral condition without angular deflection.

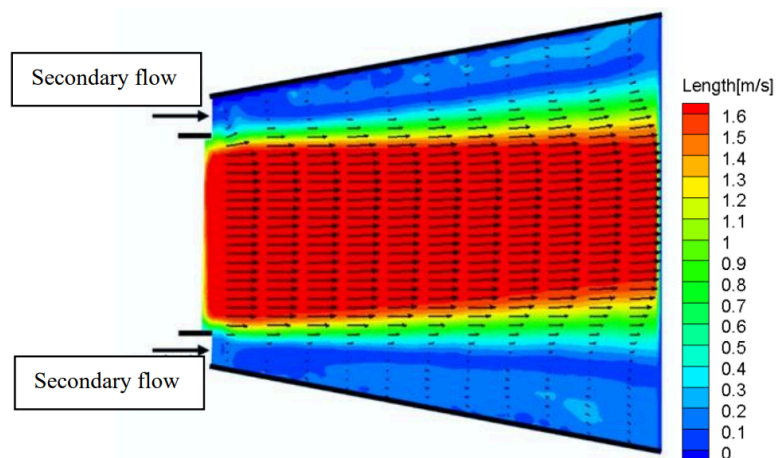


Figure 2. Velocity diagram of the flow field when the jet is neutral

3.2. Deflected main jet adhering to the wall

Figure 3 shows the center-plane velocity field when the upstream external-fluid inlet is closed. The main jet deflects upward and attaches to the upper wall. A low-velocity recirculation cell develops in the near-wall region on that side. On the opposite (lower) side, the shear layer thickens steadily from the leading edge to the trailing edge. In the trailing-edge region, fluid on the upper side glides downstream along the wall without forming notable structures, whereas on the lower side reverse suction drives fluid from the trailing edge back toward the leading edge within the near-wall layer.

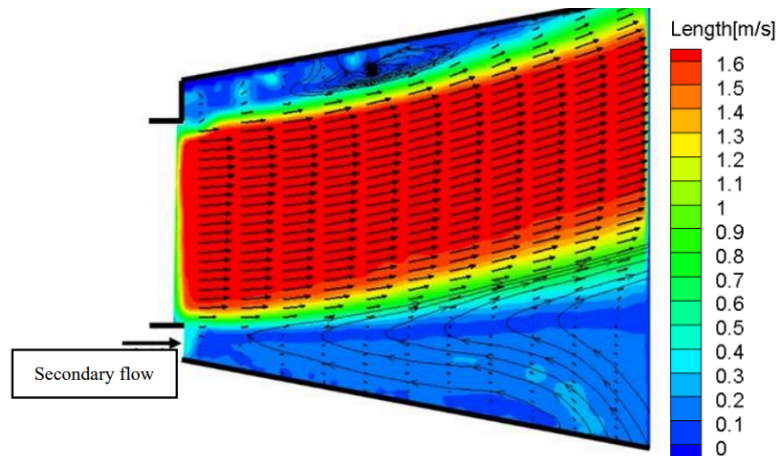


Figure 3. Velocity diagram of the flow field when the jet deflects and attaches to the wall

4. Conclusions and outlook

4.1. Summary

This work uses CFD simulations to examine the flow physics of an underwater passive fluidic thrust vectoring nozzle.

- (1) Flow field characteristics in the steady-state neutral condition

In the neutral state the jet issues straight downstream along the centerline.

- (2) Flow Field Characteristics under Steady-State Deflected Wall-Following Conditions

When the upstream external-fluid inlet is closed, the resulting pressure asymmetry pulls the jet toward the upstream wall, producing steady wall-following deflection.

4.2. Future prospects

The simulations reported here provide an initial picture of the nozzle behavior, but several open questions warrant further investigation:

- (1) Comparative studies of additional aspect ratios
- (2) Numerical Simulation of Transient Flow and Hysteresis Characteristics
- (3) Influence of Control Parameters on Deflection Performance
- (4) Integrated Simulation of the Nozzle and Vehicle
- (5) Experimental Validation

References

- [1] Mao, H., Ji, R., Lin, J., Yuan, J., & Zhao, Z. (2022). Research progress on vector pump-jet propulsion for underwater vehicles. *Guangdong Shipbuilding*, 41(1), 21-23+27.
- [2] Shi, S., Huang, X., Rao, Z., & Hua, H. (2022). Study on force spectrum characteristics of pump-jet propulsion systems under turbulent inflow. *Chinese Journal of Naval Architecture*, 17(1), 1-10. DOI: 10.19693/j.issn.1673-3185.02249.
- [3] Olsen, R. E. (1962, November 28). Reattachment of a two-dimensional compressible jet to an adjacent plate. *ASME Symposium on Fluid Jet Control Devices*.
- [4] Comparin, R. A., et al. (1962, November 28). On the limitations and special effects of fluid jet amplifiers. *ASME Symposium on Fluid Jet Control Devices*.
- [5] Warren, R. W. (1962, November 28). Some parameters affecting the design of bistable fluid amplifiers. *ASME Symposium on Fluid Jet Control Devices*.