

Article

A Numerical Investigation into the Thrust Characteristics of the RAS-HA-X25 Autonomous Underwater Vehicle Through CFD-Based Simulation

Aleksander Grm ^{1,2} , Marko Peljhan ³, Roman Kamnik ⁴ , Matej Dobrevski ⁵ , Dominik Majcen ⁴ 
and Andrej Androjna ^{1,*} 

¹ Faculty of Maritime Studies and Transport, University of Ljubljana, 1000 Ljubljana, Slovenia; aleksander.grm@fpp.uni-lj.si

² Faculty of Mechanical Engineering, LECAD, University of Ljubljana, 1000 Ljubljana, Slovenia

³ MAT Systemics, University of California, Santa Barbara, CA 93106, USA; peljhan@ucsb.edu

⁴ Faculty of Electrical Engineering, University of Ljubljana, 1000 Ljubljana, Slovenia; roman.kamnik@fe.uni-lj.si (R.K.); dominik.majcen@fe.uni-lj.si (D.M.)

⁵ Faculty of Computer and Information Science, University of Ljubljana, 1000 Ljubljana, Slovenia; matej.dobrevski@fri.uni-lj.si

* Correspondence: andrej.androjna@fpp.uni-lj.si

Abstract

The rapid development of Autonomous Underwater Vehicles (AUVs) has increased the demand for propulsion systems that balance thrust density, hydrodynamic efficiency, and acoustic discretion. This study presents a comprehensive numerical investigation of the performance of the Blue Robotics T500 thruster, embedded within the RAS-HA-X25 AUV's internal conduit. Using transient Computational Fluid Dynamics (CFD) within the OpenFOAM framework, this research assesses the propulsive characteristics of the thruster across six distinct outlet geometries, including convergent jet nozzles and multi-lobed “daisy” configurations. To improve computational efficiency for parametric design, a calibrated actuator disc model was developed and validated against resolved-rotor simulations, revealing a 15 % discrepancy attributed to tip leakage and hub vortex effects. Results show that at the operational advance ratio ($J = 0.167$), the 60 mm convergent nozzle is the optimal configuration for maximising thrust, achieving a peak net thrust of 42 N. In contrast, the daisy-type lobed geometries, while causing a 50 % reduction in absolute thrust compared to a standard cylindrical pipe, significantly homogenise the exit-plane velocity distribution and reduce swirl intensity. These findings indicate that lobed terminations provide a viable mechanism for reducing hydroacoustic signatures, offering a strategic “stealth” advantage for low-observable underwater platforms where acoustic discretion is prioritised over pure thrust density. This study establishes a robust methodology for optimising embedded propulsion modules in next-generation autonomous and hybrid underwater vehicles.



Academic Editor: Zhiguang Feng

Received: 26 February 2026

Revised: 17 March 2026

Accepted: 19 March 2026

Published: 24 March 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

Keywords: propeller; duct; hydrodynamics; BlueRobotics T500 Thruster; outflow nozzle

1. Introduction

The rapid expansion of marine robotics has made Autonomous Underwater Vehicles (AUVs) indispensable for subsea exploration, environmental data collection, defence applications, and offshore industrial operations [1–3]. As mission profiles become more sophisticated, with a focus on compact and acoustically stealthy vehicle footprints and

extended operational endurance, the need for high-precision motion control and optimal hydrodynamic integration has increased. Central to achieving these objectives is the propulsive efficiency of the vehicle's thrusters, which must provide reliable performance across a wide range of operating conditions [4].

For survey style AUVs, such as the Pirajuba[5], and the RAS-HA-X25 test platform examined in this study, integrating thrusters within the vehicle's hull, commonly known as embedded or tunnelled propulsion, provides significant advantages for mechanical protection and streamlined hydrodynamic profiles [6]. However, this configuration introduces complex internal flow dynamics that differ markedly from open-water propeller performance [7]. The confinement of flow within a conduit alters the pressure distribution across the rotor blades and requires a thorough understanding of the axial momentum recovery at the pipe termination [8].

Previous research has provided a solid foundation for understanding AUV propulsion hydrodynamics through both experimental [9] and numerical methods, described by Battacharyya et al. [10] and the comparison between panel code and Reynolds-averaged Navier-Stokes (RANS) code by Baltazar et al. [11]; a notable example is also presented by Haoda Li et al. [12] on propeller thrusters for achieving high manoeuvrability and energy-efficient motion. Early studies examined the integrated interaction between hull and propeller forces [6], while later work expanded these perspectives to include high-fidelity Computational Fluid Dynamics (CFD) analyses of drag, manoeuvrability, and hydrodynamic coefficients [13–20]. These studies validated CFD methods for predicting underwater vehicle performance and highlighted the critical roles of flow separation, turbulence, and propulsor–hull interactions.

Despite these advances, current research in AUV simulation often relies on simplified numerical models to predict vehicle behaviour. For example, simulators developed for the BlueROV2 platform, such as those discussed by Einarsson [21] and von Benzon et al. [22], frequently use Ordinary Differential Equation (ODE)-based models to approximate thruster forces. Although these models are computationally efficient for control system design, their accuracy is often limited by the fidelity of the underlying thruster data [23]. This limitation is particularly evident in ducted and thrust-vectoring systems, which have been the focus of recent research due to their superior static thrust and flow-conditioning capabilities [20,24–29]. These studies show that the performance of the ducted propeller is highly sensitive to the geometry of the nozzle, the loading of the blade, and the recovery of swirls.

To bridge the gap between low-fidelity, simplified models and physical reality, this study uses the Blue Robotics T500 thruster, Bluerobotics, Torrance, CA, USA as a benchmark. To validate our numerical framework, we apply the foundational principles of fluid mechanics and aerodynamics established by Halliday et al. [30] and Houghton and Carpenter [31], as well as AUV thruster fault diagnosis based on the comprehensive model by Jiang et al. [32]. Furthermore, to support computational efficiency during parametric studies, actuator disc models have become prominent [19,20,33,34]. We adopt the Actuator disc Theory (ADT) proposed by Sørensen [35], which represents the propeller as a mathematical pressure jump. This approach significantly reduces computational overhead while maintaining the essential physics of the induction field, provided the model is carefully calibrated against actual hardware performance to account for discrepancies in manufacturer-provided geometries.

Non-conventional designs, such as magnetic-coupling thrusters [36,37], parallel-manipulator vectored thrusters [38,39], and synthetic-jet steering mechanisms [40], reflect growing interest in innovative propulsion configurations for small, streamlined I-AUVs [41,42]. However, the influence of internal-duct exit geometry—specifically the trade-off between thrust-optimised convergent nozzles and multi-lobed, acoustically bene-

ficial “daisy” structures [43–45] remains comparatively underexplored under realistic AUV operating conditions.

This work addresses this gap by conducting a comprehensive CFD-based investigation of the T500 thruster installed inside a cylindrical tube with six distinct outlet geometries. The objectives of this research are threefold:

1. To quantify the influence of tube-exit geometry on thrust production and axial momentum recovery in embedded thruster systems.
2. To assess the accuracy of actuator disc approximations in internal-duct flow environments characterised by confinement, swirl, and unsteady vortex dynamics.
3. To identify design trade-offs between convergent nozzles and acoustically favourable lobed geometries at operational AUV speeds.

By correlating transient CFD results with analytical momentum theory, this study aims to provide new insights into the hydrodynamic behaviour of enclosed thruster systems. It may serve as a strong basis for optimising propulsion modules for next-generation autonomous and hybrid underwater vehicles.

The overall structure of the manuscript is as follows. Section 2 provides a detailed description of the problem, including the geometric characteristics of the T500 thruster, the implementation of the actuator disc model, and the specifications of the computational domain and boundary conditions. Section 3 presents the numerical results, covering the baseline characterisation of the thruster, validation of the propeller disc approximation, and a comparative performance analysis of the six distinct outlet geometries. Section 4 discusses the observed hydrodynamic trade-offs between convergent and lobed designs, and evaluates the fidelity of the transient simulation framework. Section 5 synthesises the main findings and offers engineering guidance for selecting optimal nozzle configurations tailored to specific AUV mission profiles.

2. Materials and Methods

The resolution of the problem encompassed two primary objectives. First, the evaluation of thrust and torque on the original T500 thruster equipped with a duct; and second, the assessment of thrust for the T500 thruster installed within a cylindrical tube. In the latter configuration, the tube was terminated with six distinct end geometries, which were systematically analysed to determine their hydrodynamic influence.

Figures 1 and 2 show a 3D model of a T500 thruster and a T500 thruster within the duct. The green parts indicate the rotating components, including the hub and blades.

The investigation of the ducted thruster began with an examination of the performance characteristics of the propeller. Characteristic curves were generated at the maximum motor rotational speed, while the advance velocity and the inflow velocity approaching the propeller were varied. This enabled identification of thrust and torque trends under different inflow conditions, which is critical for validating propeller efficiency in constrained flow environments.

Furthermore, for the thruster enclosed within a tube, the flow field was computed using a propeller disc model instead of the full thruster geometry. The tube geometry with various exit terminations is shown in Figure 3. This simplification significantly reduced computational cost while preserving the essential momentum exchange characteristics. The propeller disc parameters were derived from the previously generated performance characteristic curves for the T500 thruster in a duct configuration.

Consequently, for each tube termination type, thrust performance was evaluated across a range of advance velocities, with the propeller operating at maximum rotational speed to ensure consistency in the comparative analysis. The six distinct termination configurations are illustrated in Figures 4 and 5. This approach enabled a systematic

assessment of hydrodynamic interactions between the propeller-induced flow and the tube-end geometry while maintaining accurate thrust predictions.

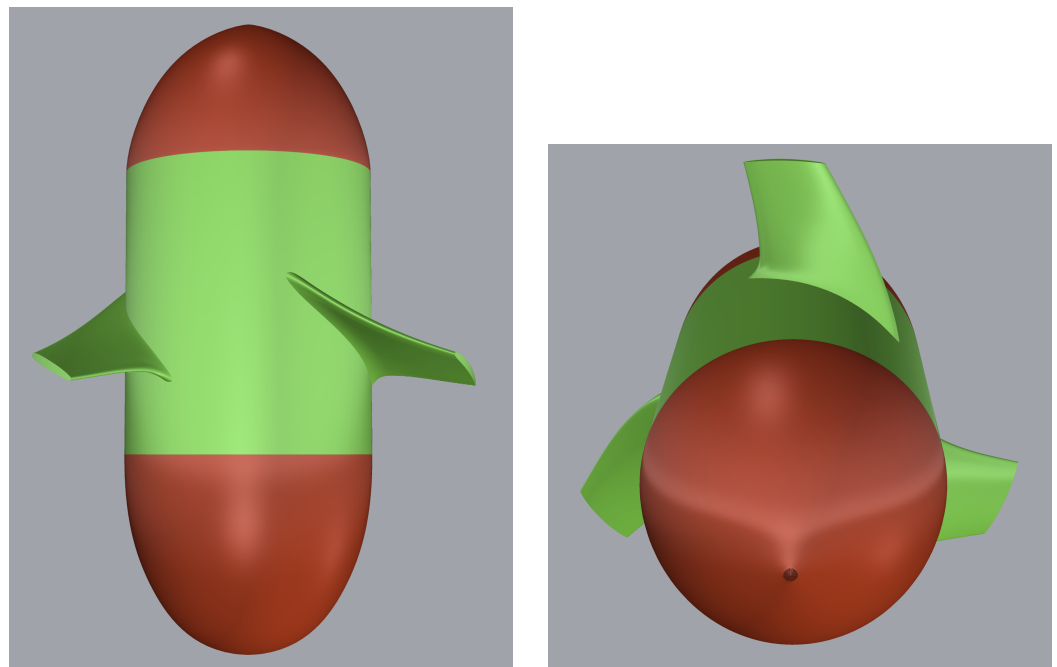


Figure 1. 3D model of the T500 thruster. The green zone indicates the rotating part, and the red zone indicates the non-rotating parts.

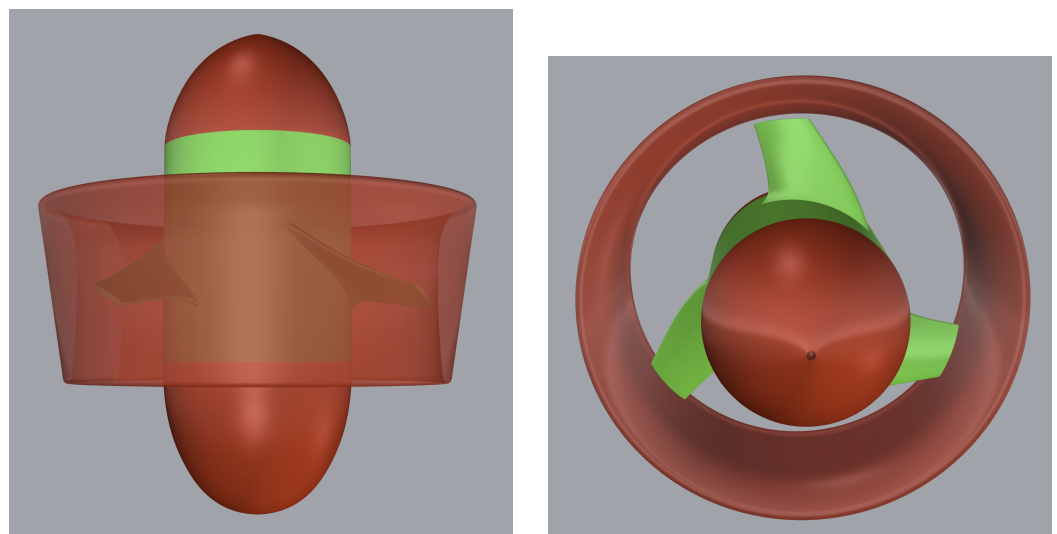


Figure 2. 3D model of the T500 thruster enclosed within a duct. The green zone indicates the rotating part, and the red zone indicates the non-rotating parts.

Before describing each geometry, it is important to note that the entire CFD analysis was performed using the OpenFOAM platform. The computational mesh was generated with the SnappyHexMesh utility, which enables automated creation of body-fitted meshes for complex geometries. This approach ensured adequate resolution near the propeller and duct surfaces while maintaining mesh quality in the far-field region, thereby supporting accurate prediction of flow characteristics.

2.1. Analysis of the T500 Thruster Enclosed Within a Duct

The propulsion system rigorously analysed in this investigation is the T500 model, engineered by BlueRobotics. The T500 is housed within a nozzle assembly which,

upon preliminary inspection, conforms to the geometric characteristics of a standard 19A duct profile [4].

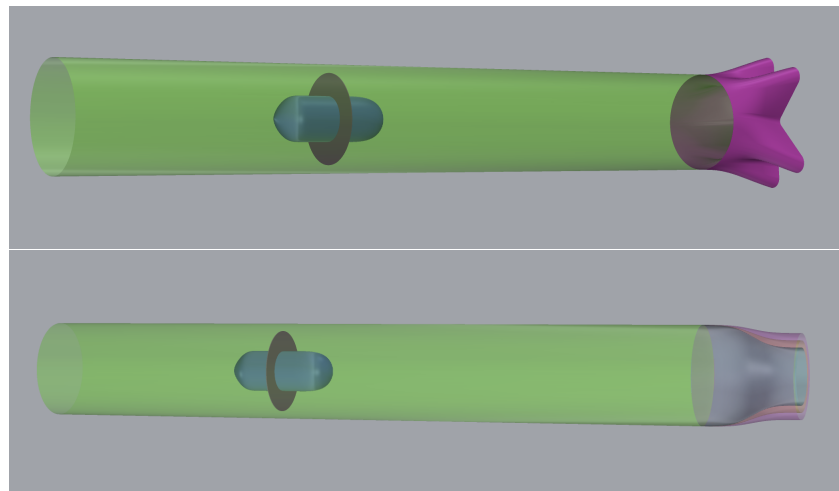


Figure 3. Configuration of propeller disc in tube or pipe with two different pipe ends.

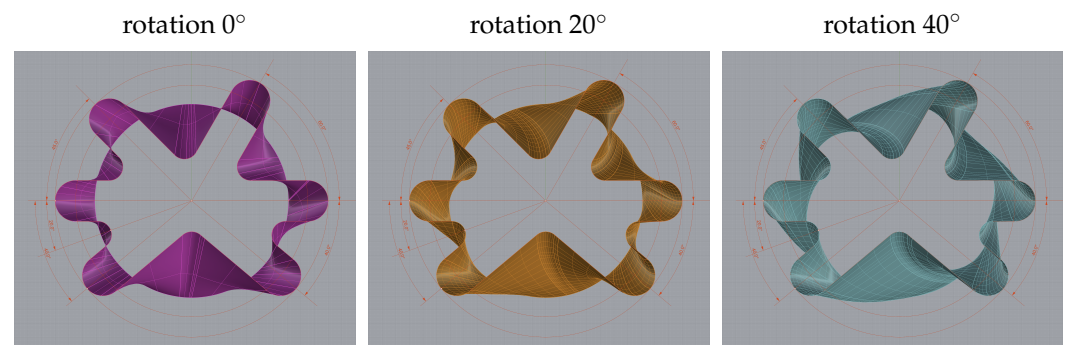


Figure 4. Pipe ends with a daisy-lobed nozzle for various angles of rotation.

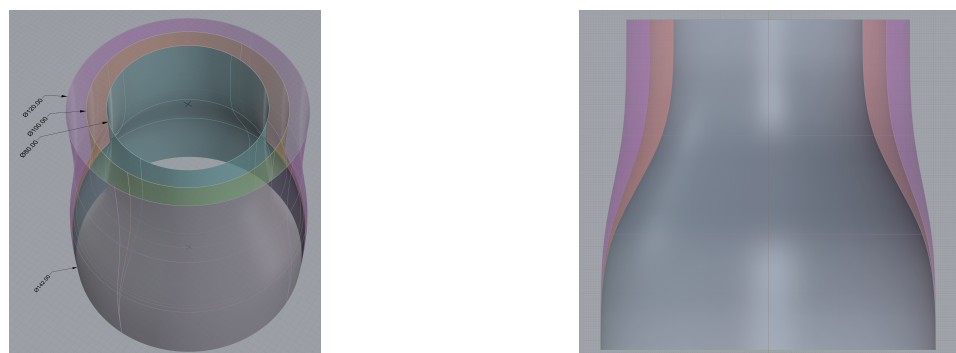


Figure 5. Pipe ends with convergent nozzle types for different outlet diameters.

To evaluate the mechanical performance of the T500, an experimental bollard-pull test was conducted over a range of angular velocities. The empirical data from these trials are documented in the manufacturer's technical repository (<https://bluerobotics.com/store/thrusters/t100-t200-thrusters/t500-thruster/>, accessed on 15 March 2026). These findings indicate that at a peak rotational velocity of approximately 3000 RPM in the ahead direction, the thruster generates a force of about 160 N.

For this analysis, a detailed three-dimensional model of the entire thruster and its duct was developed. The propeller blades were extracted from the 3D model provided by BlueRobotics in the technical specifications section of the thruster's webpage. Figures 1 and 2

show the 3D representation of the thruster and its duct. Components shown in green indicate rotating parts, while those in red represent non-rotating components. This colour coding was also used to define rotating and stationary zones in the simulation. For reference, Table 1 summarises the principal dimensions of the T500 ducted thruster.

Table 1. T500 ducted thruster dimension.

Part	Dimension
Hub radius	31 mm
Blade tip radius	57 mm
Rotating hub part length	71 mm
Complete hub length	160 mm
Duct inner radius	59 mm
Duct overall length	59 mm

2.1.1. Numerical Method

As mentioned earlier, the entire analysis was conducted using the OpenFOAM platform, version 13. Due to the inherently turbulent nature of the flow, the $k-\omega$ SST turbulence model was used. When evaluating the turbulence characteristics, the maximum velocity was recorded at the propeller tip, reaching approximately 18 m/s. At this location, the characteristic length corresponds to the propeller tip chord, approximately 17 mm. Based on these parameters, the Reynolds number is estimated to be around 3×10^5 , confirming that the flow in this region is fully turbulent and therefore requires the use of a turbulence model.

The calculated Reynolds number confirmed the fully turbulent nature of the flow and influenced the selection of the turbulence model and the mesh refinement strategy. The $k-\omega$ SST model was selected as it is particularly suitable for flows with strong adverse pressure gradients and rotating machinery such as thrusters. This model requires accurate near-wall treatment, which was achieved by refining the mesh to maintain y^+ values within the recommended range of 30–300 for wall-function-based simulations. The combination of the $k-\omega$ SST model and wall functions provided an optimal balance between numerical accuracy and efficiency for this analysis.

Figure 6 shows the results for the rotating component of the T500 thruster, illustrating the velocity magnitude (left) and the corresponding y^+ values (right). As shown in the figure, the mesh was designed to ensure that the y^+ values fully meet the requirements of the wall-function turbulence model. A detailed view of the mesh structure, including boundary layer refinement, is provided in Figure 7. In this case, the mesh consisted of approximately 14 million cells.

Computational Domain and Boundary Conditions

The computational domain consists of a cylindrical volume, as shown in Figure 8. The inflow boundary is located at the proximal end of the cylinder (indicated by the blue disc), while the outflow boundary is at the distal end (represented by the red disc). The yellow circumferential surface denotes the slip exterior wall. In this numerical configuration, the slip condition constrains the fluid velocity vectors to remain parallel to the boundary, ensuring an undisturbed flow field.

The longitudinal extent of the domain is approximately 2 m, which is twelve times the characteristic length of the thruster, with the propulsion unit positioned near the centre of the domain. The domain diameter is approximately 1 m, equivalent to roughly six times the thruster length. These spatial dimensions follow established Computational Fluid Dynamics (CFD) best-practice protocols for marine propulsor simulations. Specifically,

these guidelines require the inlet boundary to be placed at least five to six thruster diameters upstream, with the outlet boundary ten to twelve diameters downstream, to mitigate blockage effects and enable accurate pressure recovery. Additionally, a lateral clearance of at least five diameters is maintained to prevent unwanted confinement effects during the simulation.

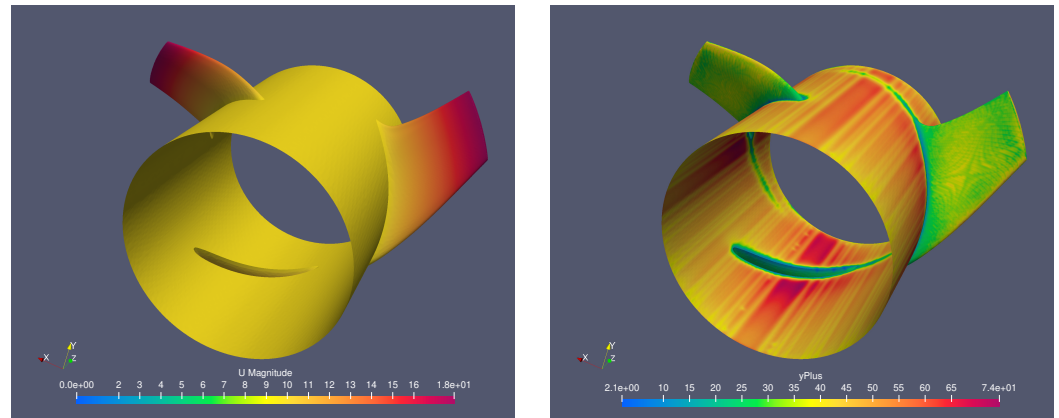


Figure 6. Simulation results of velocity magnitude and y^+ for the rotating part with an angular velocity of 3000 RPM. The left figure shows the velocity magnitude of the blades, while the right figure displays the corresponding y^+ values in the operating regime.

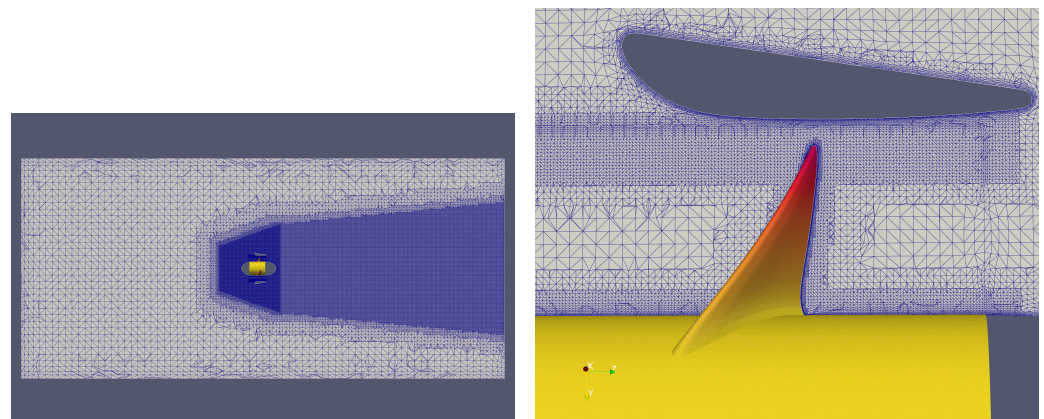


Figure 7. Details of the mesh for the T500 thruster duct problem, including the boundary layer mesh.

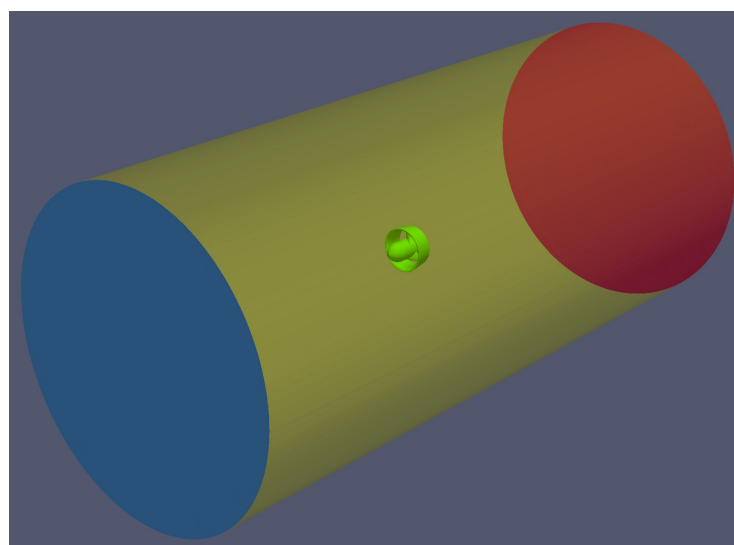


Figure 8. Computational domain of the ducted problem. Blue indicates the inflow face, red indicates the outflow face, and the cylinder face is the symmetry face.

The no-slip boundary condition was applied to the duct surface and to the front and rear faces of the thruster hub (red zones in Figure 1). At the inflow boundary, several inlet velocities were specified:

$$v_{\text{inflow}} = \{0.1, 0.2, 0.3, 0.5, 1.0, 1.5, 2.0, 2.5, 3.0, 3.5, 4.0, 4.5, 5.0\} \text{ m/s} \quad (1)$$

using the `flowRateInletVelocity` condition for velocity and `zeroGradient` for pressure. The outflow boundary employed `pressureInletOutletVelocity` for velocity and `entrainmentPressure` for pressure. On the outer cylindrical surface of the domain, a slip boundary condition was imposed to minimise wall interference effects.

Computational Mesh

To ensure that the numerical results were independent of spatial discretisation, a systematic grid convergence study was conducted. The computational mesh was generated using the `SnappyHexMesh` utility, enabling the creation of body fitted meshes with targeted refinement in high-gradient regions, such as the propeller tips and duct interfaces. For the resolved-rotor simulations, three mesh densities—coarse, medium, and fine—were assessed to monitor the stability of the integral performance metrics.

The computational mesh surrounding the thruster region is shown in Figure 7. To ensure an undisturbed flow field and minimise blockage effects in line with established CFD best practice protocols, the domain was defined with a longitudinal extent of twenty times the thruster's characteristic length ($20L$) and a width of $10L$. The inlet boundary was positioned $7L$ upstream of the propulsion unit, while the outlet was placed $13L$ downstream to allow for accurate pressure recovery. The lateral clearance was set at $5L$ both above and below the thruster to prevent confinement effects during the simulation.

As shown in Figure 7, the mesh density was highest near the thruster, where the smallest cell size was reduced to one tenth ($1/10$) of the reference cell size used in the far field region. To verify that the numerical solution was independent of the spatial discretisation [46], three mesh types were constructed: a high-fidelity “fine” mesh, a “medium” mesh with a density of two-thirds ($2/3$) that of the fine mesh, and a “coarse” mesh at one-half ($1/2$) the density [46]. This hierarchical refinement was systematically controlled by the reference cell size, providing a robust framework for grid convergence assessment.

The high-fidelity fine mesh selected for the T500 ducted thruster analysis comprised approximately 14 million cells. This configuration included extensive boundary layer refinement to maintain y^+ values within the range 30–300, ensuring that the $k-\omega$ SST turbulence model could accurately resolve the flow in the logarithmic wall region. In contrast, the mesh for the propeller disc configurations was optimised to approximately 7.5 million cells. This reduction in discretisation complexity was essential for achieving computational efficiency during the parametric evaluation of the six different exit geometries. The main mesh data are shown in Table 2.

The convergence of the primary hydrodynamic parameters, specifically thrust force and torque, was confirmed as residuals for all transport variables reached the 10^{-4} threshold. Even in lower velocity regimes where local instabilities occurred, the asymptotic behaviour of these integral quantities indicated that the global flow features were sufficiently captured, thereby validating the selected mesh resolutions for robust engineering assessment.

Multiple Reference Frame (MRF) Simulations and Convergence Behaviour

For the computation of the rotating region, the OpenFOAM Multiple Reference Frame (MRF) approach was used. This method defines a rotational sub-domain, which in this case was a cylindrical region with a radius of approximately 58 mm, located between the inner

surface of the duct and the tips of the thruster blades. The length of this cylinder extends from the front to the rear of the thruster hub. The angular velocity was set to the maximum operating speed of the T500 thruster, 3000 RPM, for all cases.

Table 2. Mesh size dimensions.

Type	Size of a Reference Cell	Number of Cells
T500 thruster		
Coarse	50 mm	3.5 M
Medium	33 mm	6.7 M
Fine	25 mm	14 M
Propeller disc		
Coarse	40 mm	2 M
Medium	27 mm	5.5 M
Fine	20 mm	7.5 M

The velocity boundary condition for the thruster blades and rotating hub was specified as MRFnoSlip, while the pressure boundary condition was set to zeroGradient. All simulations were performed in steady-state mode, with a maximum of 4000 iterations and a residual convergence criterion of 10^{-4} for all variables (U , p , k , ω , and v_t). This threshold was chosen as a practical balance between computational efficiency and numerical accuracy, as residuals below 10^{-4} typically indicate negligible changes between iterations and sufficient stability for engineering predictions.

For low inlet velocities (below 1 m/s), full convergence was not achieved within the iteration limit; however, residuals decreased to approximately 3.5×10^{-4} by the 4000th iteration, which was considered acceptable since global performance metrics, such as thrust and torque, were the primary focus. Achieving lower residuals would require significantly more iterations and computational time, with minimal impact on these integral quantities. For inflow velocities equal to or above 1 m/s, the convergence criterion was satisfied well before the iteration limit, typically within 2000 iterations, confirming the robustness of the numerical setup under higher flow conditions.

2.2. Analysis of the T500 Thruster Enclosed Within a Tube

In the second phase of this investigation, the configuration in which the thruster is enclosed within a cylindrical conduit was analysed. This arrangement reflects the architectural design of our bespoke underwater vehicle, which uses a dual-thruster propulsion system, as shown in Figure 3.

To evaluate the hydrodynamic conditions within the conduit, the thruster was represented using an actuator disc model, as shown in Figure 9. Following the methodology outlined in the preceding subsection, the characteristic performance profiles of the T500 thruster were obtained, including the non-dimensional thrust coefficient K_T , torque coefficient K_Q , and propulsive efficiency η . The complete empirical dataset is provided in Section 3. These performance curves were generated by applying cubic B-spline interpolation to the numerical data across a range of advance ratios J , based on the inlet velocities defined in Equation (1). Using these coefficients, the actuator disc model enables simulation of thruster dynamics without resolved blade geometry.

The reduction in computational overhead provided by this approach is crucial; it greatly simplifies the discretisation process and reduces the global cell count. This is essential for achieving the required computational efficiency within our hardware constraints. Managing mesh density is particularly important, as the simulation matrix requires evaluating six distinct exit geometries at three inlet flow velocities.

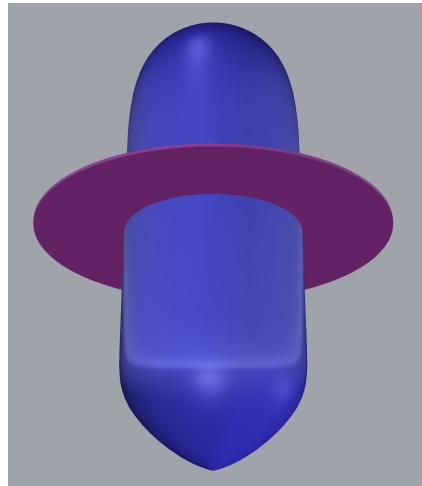


Figure 9. Actuator disc representation of the propeller with a central hub. The disc approximation replaces the propeller blades to simplify thrust analysis.

2.2.1. Thruster–Tube Geometry

To assess the thrust performance of a tube-mounted thruster, the propeller is modelled using the actuator disc approximation, as shown in Figure 9. This idealisation replaces the discrete blade geometry with a uniformly loaded disc, simplifying the analysis of momentum exchange between the incoming flow and the rotor. The disc area corresponds to the annular region bounded by the inner and outer radii of the propeller blades, representing the effective swept surface that transfers momentum.

The disc–hub assembly is positioned within a cylindrical tube, through which water enters at a velocity equal to the drone’s translational speed, replicating the relative motion between the vehicle and the surrounding fluid. Downstream, the tube ends in several alternative outlet geometries, as shown in Figure 3. These configurations are introduced to examine their influence on flow acceleration, pressure recovery, and overall thrust generation. This approach provides a controlled framework for evaluating hydrodynamic interactions and serves as a baseline for comparative thrust analysis across different outlet designs, see Table 3.

Table 3. Dimensions of actuator disc within a tube system.

Part	Dimension
Hub radius	31 mm
Blade tip radius	57 mm
Rotating hub part length	71 mm
Complete hub length	160 mm
Duct inner radius	59 mm
Duct overall length	59 mm

3. Results

The research findings were categorised into three main subsections: the hydrodynamic analysis of the thruster, the derivation of characteristic propeller performance curves, and the evaluation of propulsive thrust within various duct geometries. The final subsection was further divided into two distinct nozzle configurations: multi-lobed “daisy” geometries and convergent jet nozzles.

In the initial phase, a hydrodynamic analysis of the thruster was conducted, in which the performance characteristics of the Blue Robotics T500 thruster were calculated through high-fidelity Computational Fluid Dynamics (CFD). Turbulent wall functions were em-

ployed within the CFD framework, and the computational mesh was designed to maintain the calculated y^+ values at or above 30. Characteristic performance graphs were produced to compare the thrust coefficient (K_T), torque coefficient (K_Q), and efficiency (η) as a function of the advance ratio (J). It was found that the T500 propeller achieved its maximum efficiency of slightly over 10 % at an advance ratio of $J = 0.5$. The operational regime for the specific Autonomous Underwater Vehicle (AUV) was set at a translational speed of 1 m/s, corresponding to $J = 0.167$ in this study.

The subsequent analysis focused on the performance characteristics of an actuator disc model, used as a substitute for the resolved T500 thruster geometry. The actuator disc enabled a significantly simplified mesh representation, while maintaining the integrity of the objective results. This simplification substantially reduced computational requirements, allowing for a more detailed and comprehensive analysis of the global model. Although the surrogate actuator disc reduced processing times, the fundamental analytical core was preserved, ensuring that the comparative value of the results was maintained.

In the third subsection, the fluid flow dynamics were evaluated for various outlet nozzle designs. The investigation included several exit configurations, the first of which featured a “daisy-shaped” geometry intended to modify the thruster’s acoustic signature. For this specific topology, three variations were examined: one with an orientation parallel to the main conduit, and two with angular offsets of 20° and 40° . These rotations were intended to compensate for rotational flow swirl, functioning similarly to the flow-straightening mechanisms found in pump-jet propulsion systems, where the rotor is the primary source of swirl.

The second category of duct terminations involved convergent nozzles with exit diameters of 60 mm, 50 mm, and 40 mm. This part of the investigation aimed to quantify the deterministic relationship between nozzle contraction and the resulting increase in jet velocity under constrained internal flow conditions.

3.1. T500 Thruster Characteristics

The T500 thruster was analysed over a comprehensive range of inlet velocities, as defined by Equation (1). This enabled a comparative assessment of the computed thrust under zero-velocity conditions ($v = 0$ m/s), corresponding to the static bollard pull test for which manufacturer data are available. Figure 10 presents the resulting flow velocity field for an ambient inlet velocity of 1.5 m/s, where the maximum reaches approximately 8 m/s.

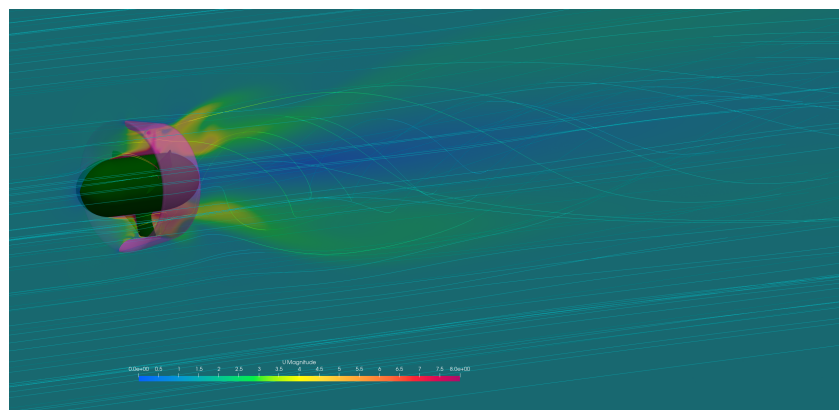


Figure 10. Computed flow velocity field for the ducted rotor configuration at an advance velocity of 1.5 m/s (colour bar: $\text{mag}(U) \in [0, 8]$ m/s).

The velocity and pressure distributions across the rotor assembly are shown in Figure 11. The velocity contours clearly delineate the kinematic boundary between the central rotating elements of the rotor and the proximal and distal stationary components (the static part of the

hub). This spatial differentiation confirms the integrity of the rotating frame of reference in the numerical model.

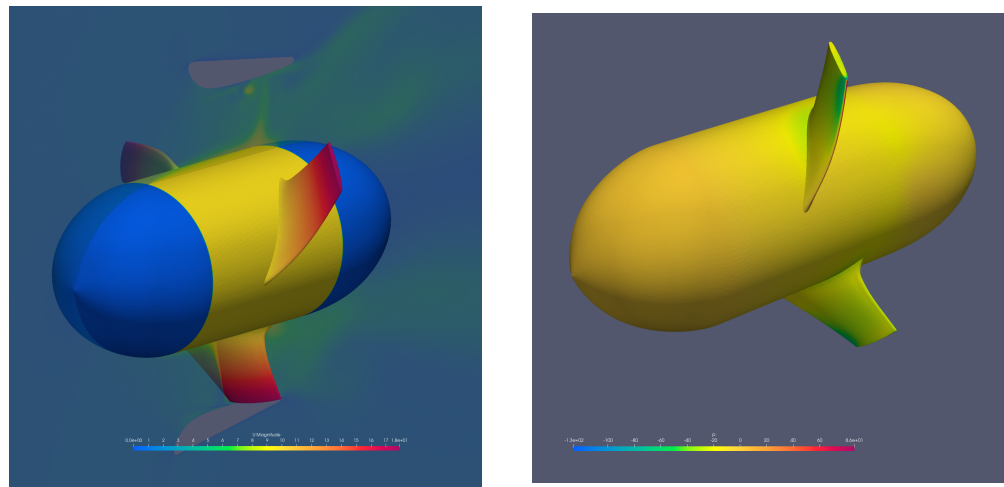


Figure 11. Distribution of the axial velocity field (**left**) and dynamic pressure (**right**) across the rotor plane at an advanced velocity of 1.5 m/s (colour bar at left: $\text{mag}(U) \in [0, 18]$ m/s, colour bar at right: $p \in [-130, 86]$ Pa).

As previously outlined in the problem definition, numerical convergence instabilities were observed at inlet velocities below 0.5 m/s. This phenomenon is primarily attributed to highly non-linear flow topologies and potential recirculation zones prevalent at low advance ratios. Despite these local instabilities, the integrated parameters—specifically thrust force and torque—remained stable throughout the iterative process. This asymptotic behaviour of the primary performance metrics suggested that the global flow features were sufficiently captured, removing the need for further modifications to the simulation parameters. Consequently, in these lower-velocity regimes, the results were considered sufficiently accurate for a robust preliminary assessment.

Conversely, at higher velocity regimes, the numerical solution demonstrated robust convergence, with residual thresholds for all transport variables consistently maintained at 10^{-4} . This high level of numerical fidelity enabled precise determination of the pressure distribution across the rotor assembly, which, in turn, allowed for calculation of the thrust force and torque demand for specific operational regimes. Figure 12 presents these characteristics for the T500 thruster, based on steady-state simulations conducted over a comprehensive range of inlet velocities at a constant rotational speed of 3000 RPM.

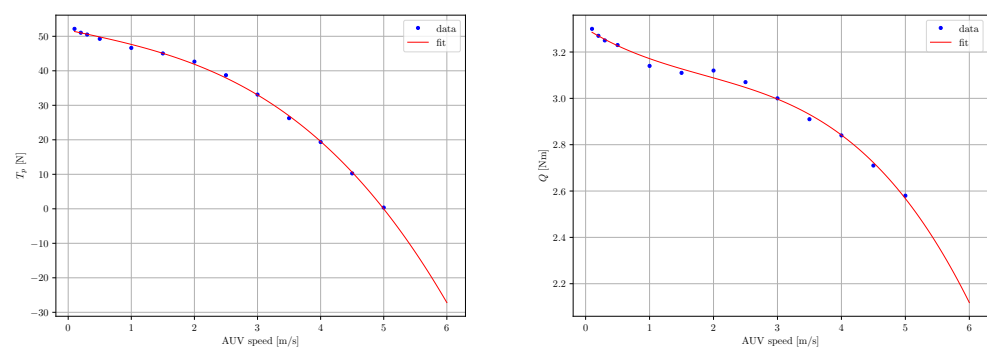


Figure 12. Thrust and torque characteristics of the T500 thruster as a function of advance velocity at a constant rotational speed of 3000 RPM. The red curves show the results of a third-order polynomial regression, illustrating performance trends across the operational envelope investigated.

As shown in Figure 12, the data points for velocity regimes below 1.0 m/s exhibit strong physical consistency and closely align with the regressed third-order polynomial functions. The coefficients for these polynomials were determined using the NumPy `polyfit` function, which applies a least-squares approach to minimise the residual sum of squares between the numerical observations and the fitted model.

The strong correlation between the numerical results and the fitted curves provides confidence in the dataset's reliability, even when formal convergence criteria were not fully met. In the context of propulsor characterisation, this alignment with the expected performance trend suggests that the underlying hydrodynamic forces were captured with sufficient accuracy for engineering assessment. Consequently, the regression model acts as an effective filter, smoothing minor numerical fluctuations and providing a reliable empirical basis for defining the actuator disc source terms within the global fluid domain.

Using the comprehensive numerical dataset, a characteristic propeller performance diagram can be produced. The performance of the T500 thruster is defined by a set of non-dimensional coefficients, which facilitate comparison of performance across different scales and operational conditions:

$$J = \frac{V_a}{nD}, \quad K_{Tp} = \frac{Tp}{\rho n^2 D^4}, \quad K_Q = \frac{Q}{\rho n^2 D^5}, \quad \eta = \frac{K_{Tp}}{K_Q} \frac{J}{2\pi} \quad (2)$$

In these expressions, J denotes the advance ratio (often called the advance number), K_{Tp} represents the non-dimensional propeller (rotor) thrust coefficient, and K_Q signifies the propeller (rotor) torque coefficient. The overall propulsive efficiency is denoted by η . Here, V_a is the speed of advance, n is the rotational frequency, D is the propeller diameter, and ρ is the fluid density.

Deriving these dimensionless parameters is essential for the subsequent development of the actuator disc model. By mapping the thruster's performance onto the J spectrum, the numerical framework can dynamically interpolate the required source terms based on the local flow conditions encountered during the underwater drone's operational envelope.

The performance characteristics of the T500 thruster are shown in Figure 13. The maximum efficiency of this thruster occurs at an advance ratio of 0.5 and is approximately 0.1. Comparable thrusters have maximum efficiencies between 0.3 and 0.35. The operational velocity of the bespoke AUV is approximately 1 m/s, corresponding to an advance ratio (J) of 0.167. The derived coefficients for K_T and K_Q will populate the lookup tables required to calculate body force source terms within the actuator disc numerical framework.

The numerical results obtained in this study were validated against empirical data provided in the manufacturer's technical repository. Specifically, the manufacturer documents performance under static conditions ($V_a = 0$ m/s), commonly referred to as a bollard pull test. Under these conditions, the reported thrust T is approximately 160 N, with a corresponding power consumption P_D of about 1 kW. The delivered power P_D is determined by the following relationship:

$$P_D = 2\pi nQ \quad (3)$$

As shown in Figure 14, the power consumption predicted by the numerical model closely matched the manufacturer's operational specifications. However, a significant discrepancy was observed in the thrust output: the calculated thrust slightly exceeded 50 N, which is about one-third of the 160 N reported by the manufacturer for static bollard pull conditions. Comparable commercial thrusters typically achieve efficiencies between 30% and 35%, whereas the maximum efficiency recorded in this study was approximately 10%, as indicated in Figure 13. As the torque coefficient was assumed correct based on

the accurate power prediction, the nearly threefold difference in the thrust coefficient (K_T) further confirmed the inconsistency of the results.

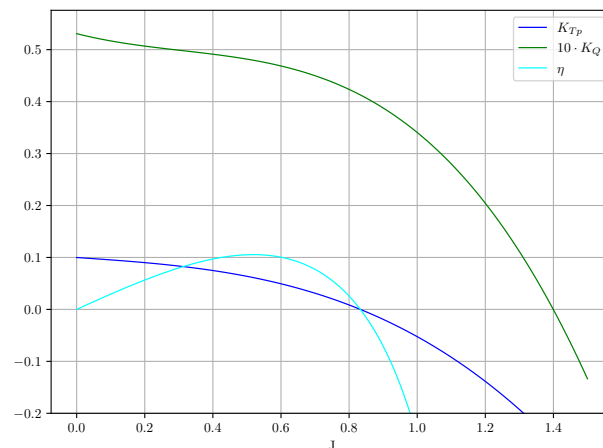


Figure 13. Non-dimensional performance characteristics of the T500 thruster. The operational velocity of the bespoke AUV, approximately 1 m/s, corresponds to an advance ratio of $J = 0.167$, which defines the primary hydrodynamic regime for this study.

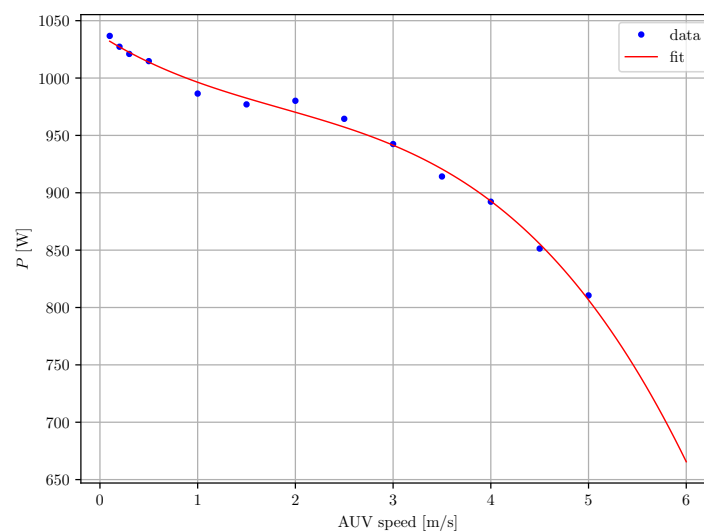


Figure 14. Distribution of the delivered power P_D for the T500 thruster across the investigated velocity range. The numerical results show a strong correlation with the manufacturer's operational specifications.

This divergence in thrust, despite the high fidelity of the power demand prediction, suggested that the 3D rotor geometry provided by the manufacturer for public release may have differed from the actual physical components used in the commercial T500 unit. Such geometrical inconsistencies are recognised as a persistent challenge in high-fidelity numerical modelling of commercial off-the-shelf (COTS) propulsion systems. In this study, these discrepancies highlighted the importance of using a comparative methodology, which allows for relative performance gains across different nozzle geometries to be isolated from systemic geometric errors.

Despite the significant discrepancy between the manufacturer's reported thrust and the numerically predicted values, it must be emphasised that the primary objective of this investigation is not to determine the absolute thrust magnitude. Instead, the study prioritises a comparative performance evaluation across various nozzle/pipe exit geometries to identify their relative hydrodynamic merits and deficiencies.

The central focus of this research is to quantify the sensitivity of the propulsive output to modifications in the pipe termination topology. By maintaining a consistent numerical framework and mesh topology across all simulations, the relative fluctuations in thrust and propulsive efficiency can be isolated from systemic errors. This approach enables a definitive assessment of which geometric configurations optimise the propulsive performance of the AUV and establishes the extent to which specific nozzle designs enhance or attenuate effective thrust relative to the baseline cylindrical duct.

3.2. Propeller Disc Characteristics

During the pipe-flow simulations, the propeller was replaced with a propeller disc to reduce the computational complexity of the problem. This substitution was implemented in OpenFOAM using the *discSource* model. A simulation of a ducted propeller disc was performed, allowing for a comparison of the results from the real propeller and the propeller disc. For the ducted propeller disc case, the computational mesh comprised approximately 7.5 million cells. Figure 15 shows the computational domain of the ducted propeller disc along with the corresponding mesh quality.

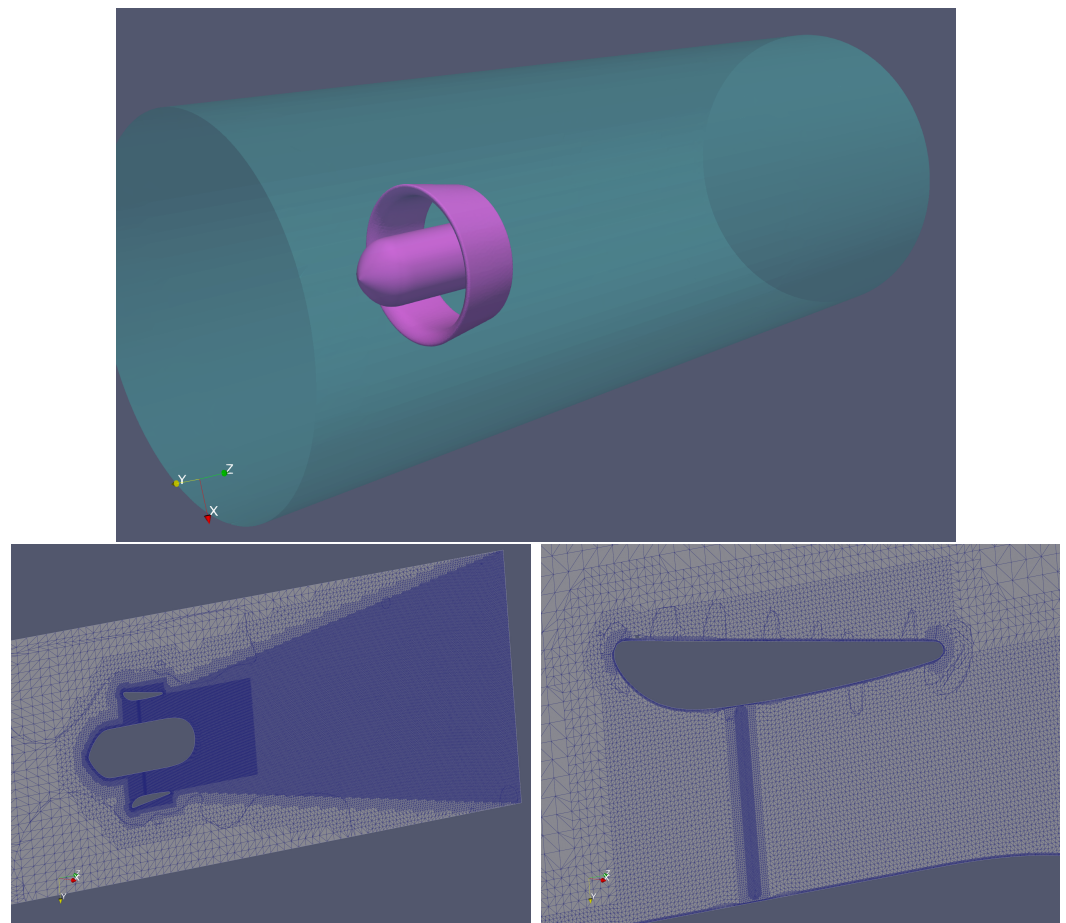


Figure 15. The upper part shows the domain configuration of the ducted propeller disc, while the lower part displays the fine mesh quality for this case.

The most challenging aspect of the study involving the substitution of the propeller with a propeller disc was the inability of the steady-state simulation to achieve convergence, which was attributed to the inherent flow characteristics within the duct when using the propeller disc model. Due to the pronounced flow unsteadiness, a transient simulation with a physical duration of 2 s was conducted to capture the complete flow development. The disadvantage of the transient approach is its substantial computational cost, requiring

approximately one day of computation on our cluster for a single simulation. Given these considerable computational demands, the use of a propeller disc as a mesh simplification was a necessary step for the full pipe-flow simulation case.

The results for the propeller disc configuration at an advance velocity of 1 m/s are shown in Figure 16. The upper panel presents the velocity magnitude field, with streamlines indicating the specific flow realisation. Compared to the resolved-rotor results in Figure 10, the flow patterns are highly similar, indicating that the actuator disc accurately captures the essential momentum exchange characteristics of the ducted system. The lower panels show the velocity vector field within the central YZ plane: the left-hand figure displays the velocity magnitudes, while the right-hand figure shows the axial velocity component (u_z). This comparative analysis confirms that the propeller disc performs as expected in replicating the induction field, providing a reliable basis for the subsequent mesh-simplified pipe-flow simulations.

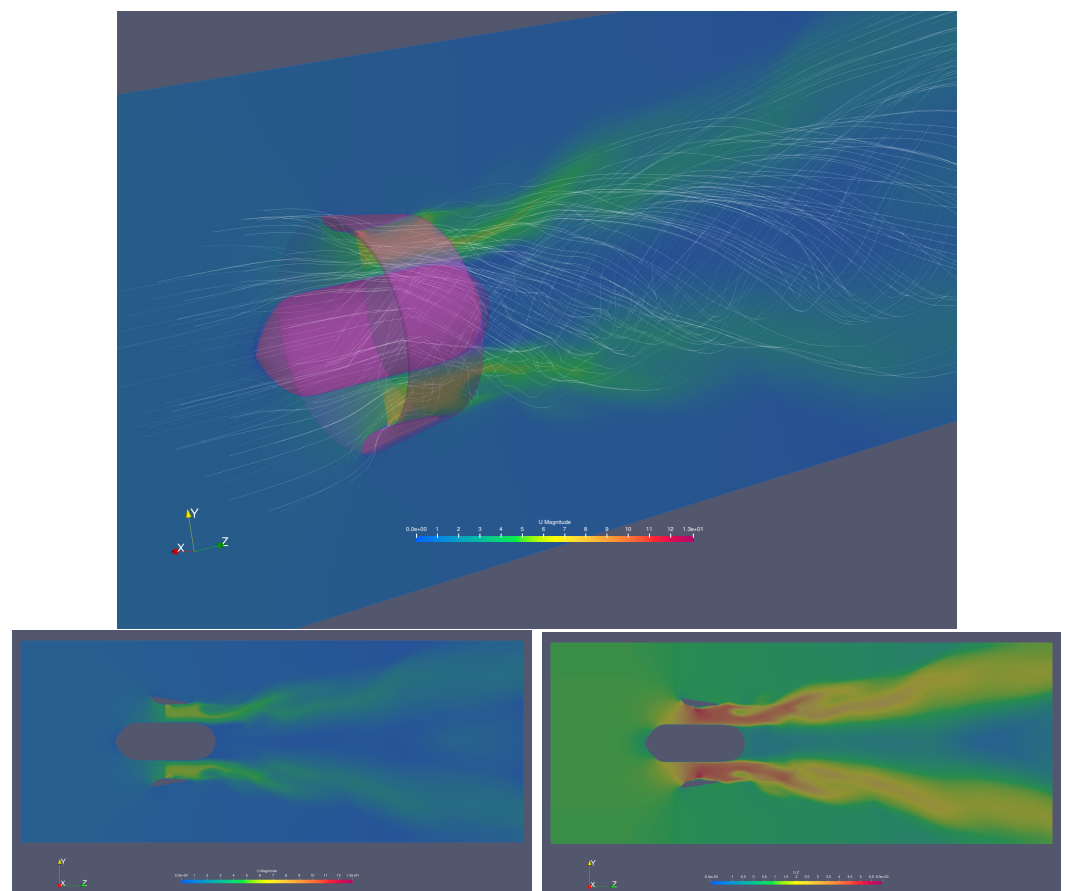


Figure 16. Solution of the ducted propeller disc for an inlet velocity of 1 m/s. The top and bottom-left panels show the velocity magnitude, while the bottom-right panel illustrates the axial velocity component U_z (bottom left: $\text{mag}(U) \in [0, 13]$ m/s, bottom right: $U_z \in [-2, 6]$ m/s).

Table 4 summarises the input data used for the propeller disc configuration. In OpenFOAM, these parameters are specified in the `constant/fvModels` file using the `discSource` keyword.

Figure 17 shows the spatial configuration of the evaluation planes used to sample the velocity vector fields and dynamic pressure distributions within the internal duct region. The median plane is located immediately downstream of the propeller disc, aligned with the mid-section of the rotor blades to capture the primary momentum exchange.

For actuator disc verification, the theoretical thrust was calculated using the methodology proposed by Sørensen [35].

$$T_{\text{disc}} = \frac{1}{2} \rho A_d (\vec{v} \cdot \vec{n})^2, \quad (4)$$

where ρ is the fluid density, A_d is the disc area, $\vec{v}$ is the velocity vector field in the plane just behind the disc (median evaluation plane at z_2), and $\vec{n}$ is the normal to the section plane. This analytical validation ensures that the numerical source terms implemented in the CFD model accurately represent the physical load distribution of the T500 rotor.

Table 4. Propeller disc configuration parameters: disc diameter $D_{\text{disc}} = 0.12$ m, hub diameter $D_{\text{hub}} = 0.06$ m, and rotational speed $n = 50$ Hz.

J	K_T	K_Q
0.10	0.0953	0.0517
0.15	0.0930	0.0512
0.19	0.0904	0.0507
0.24	0.0876	0.0504
0.29	0.0844	0.0500
0.33	0.0808	0.0496
0.38	0.0767	0.0493
0.43	0.0721	0.0489
0.47	0.0669	0.0485
0.52	0.0611	0.0479
0.57	0.0545	0.0473
0.61	0.0471	0.0466
0.66	0.0389	0.0458
0.71	0.0298	0.0448
0.75	0.0197	0.0437
0.80	0.0085	0.0424

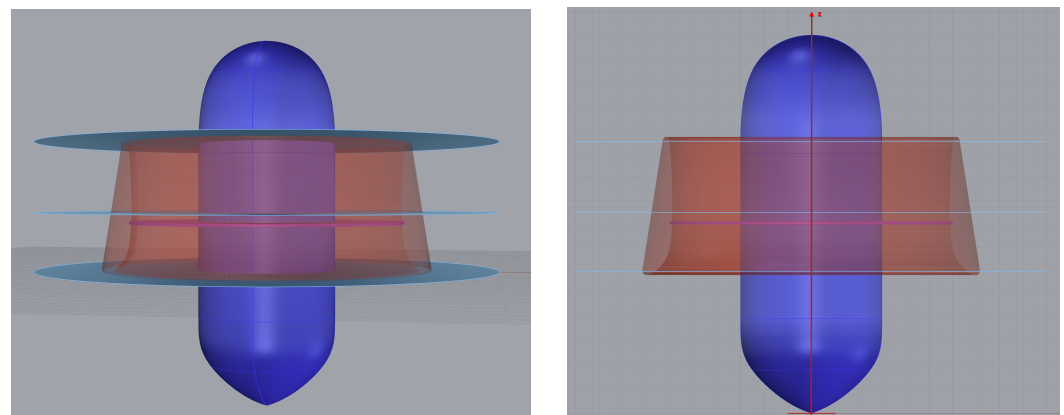


Figure 17. Spatial configuration of the evaluation planes within the duct. The coordinate origin $(0, 0, 0)$ is located at the thruster nose, with the flow directed along the positive z -axis. Planes are positioned at axial heights of $z_1 = 60$ mm, $z_2 = 85$ mm, and $z_3 = 115$ mm to monitor internal velocity and pressure distributions.

Figures 18 and 19 present the numerical results for the median evaluation plane at an advance velocity of $V_a = 1$ m/s. The computational domain is defined by a hub radius of 31 mm and a disc radius of 60 mm. The figures provide a comparative analysis of the propeller disc model, shown in Figure 19, and the thruster with the resolved rotor, shown in Figure 18. In each row, the velocity magnitude ($\text{mag}(U)$), the axial velocity component (U_z), and the dynamic pressure distribution (p) are displayed, together with their respective area-averaged values calculated over the section area (A). These averaged quantities are

then used to calculate thrust performance according to the analytical framework established in Equation (4). Comparing the solutions shown in Figures 18 and 19 in terms of magnitude, the propeller disc performs as expected.

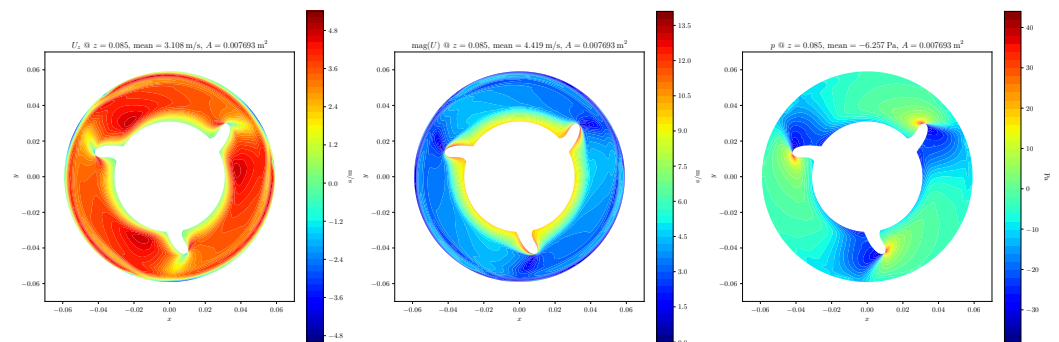


Figure 18. Results for a rotor at the mid- plane at $z = 85$ mm for a vehicle speed of $V_a = 1$ m/s. The left figure shows the velocity component in the flow direction (U_z), the middle figure shows the velocity magnitude ($\text{mag}(U)$), and the right figure shows the dynamic pressure distribution.

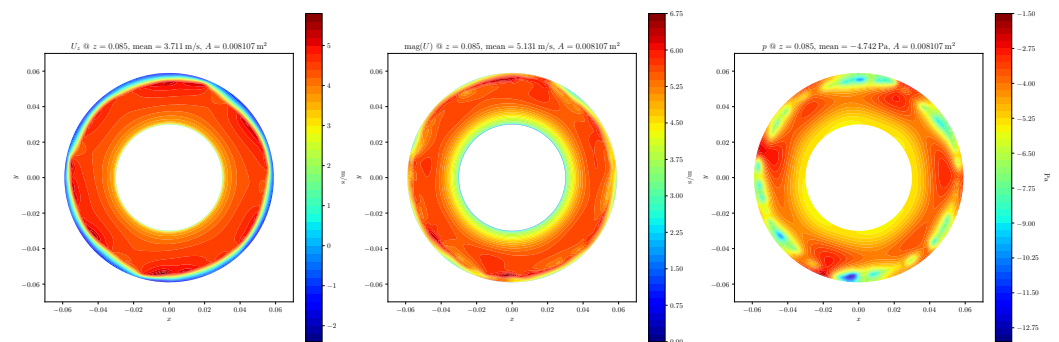


Figure 19. Results for a propeller disc at the mid- plane $z = 85$ mm with vehicle speed $V_a = 1$ m/s. The left figure shows the velocity component in the flow direction (U_z), the middle figure shows the velocity magnitude ($\text{mag}(U)$), and the right figure shows the dynamic pressure distribution.

At this operational velocity, the computed thrust for the resolved rotor case is approximately $T_p = 48$ N (as shown in Figure 12), whereas the actuator disc model yields a thrust of $T_d = 57.22$ N. Although a discrepancy of approximately 15 % exists between the two modelling approaches, it does not significantly affect the overarching objectives of this study.

The primary focus remains the comparative analysis of propulsive efficiency across different nozzle geometries. As the performance of each exit nozzle will be evaluated using equal actuator disc thrust as the source term, the current level of agreement between the resolved rotor and the simplified disc is considered sufficient. This ensures that the qualitative trends and relative performance gains identified in the nozzle study are based on a physically comparable baseline.

3.3. Propulsive Performance and Nozzle Exit Morphology

The analysis of various outlet geometries demonstrated a clear performance trade-off: convergent nozzles excelled in axial thrust generation through flow acceleration, whereas daisy-shaped lobed terminations provided superior flow uniformity and swirl reduction. This section evaluated how these distinct geometric strategies influenced the hydrodynamic efficiency of the T500 thruster at the AUV's operational advance ratio ($J = 0.167$).

In the previous section, we presented the 3D design of daisy-shaped terminations in Figure 4. The daisy-shaped terminations allowed for configurations with different rotations by rotating the geometry at the contact plane between the conduit and the end-piece. This

approach ensured that the profile of the exit surface remained constant across all variations, which greatly facilitated the post-processing of the numerical results. In contrast, Figure 5 shows the geometry of the convergent nozzle terminations. For these configurations, it was necessary to process the results specifically for the cross-sectional area corresponding to each individual diameter to quantify accurately the relationship between nozzle contraction and jet velocity enhancement.

To evaluate the propulsive efficacy of the proposed modifications, the thrust performance of each nozzle configuration was benchmarked at three discrete inlet velocities, specifically, 1.0 m/s, 1.5 m/s, and 2.0 m/s against the baseline cylindrical conduit. This comparative study focuses primarily on the mechanisms of axial thrust recovery and the deterministic influence of exit geometry on downstream wake development. Due to inherent instabilities in the flow field within the conduit, a converged stationary solution could not be obtained. Consequently, a transient numerical approach was adopted to accurately capture the time-dependent characteristics of the flow.

The simulation workflow began with a steady-state run of 1000 iterations, the results of which provided the initial conditions for the subsequent transient analysis. The transient simulations covered a total physical time interval of 2 s. To reduce the impact of initial numerical oscillations and obtain a statistically stable thrust value, a temporal averaging technique was used. The final thrust results were calculated as the mean value at three discrete time steps: 1.8 s, 1.9 s, and 2.0 s. This approach ensures that the reported values represent the quasi-steady state of the propulsor, filtering out high-frequency fluctuations that could otherwise obscure the underlying hydrodynamic performance trends, see Figure 20.

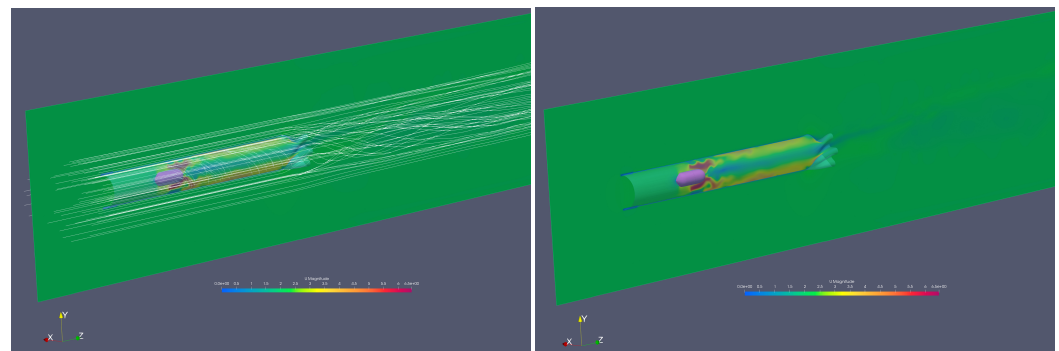


Figure 20. Result of pipe flow for an inlet velocity of 2.0 m/s with a daisy 40° nozzle (colour bar at left: $\text{mag}(U) \in [0, 6.5]$ m/s, colour bar at right: $\text{mag}(U) \in [0, 6.5]$ m/s).

The thrust force was quantified by applying the principles of Froude's Momentum Theory [30,31], adapted for internal duct flow. The governing equations for mass flow rate and propulsive thrust are defined as follows:

$$\begin{aligned} \dot{m} &= \rho A_{\text{end}} v_{\text{end}}, \\ T &= \dot{m}(v_{\text{end}} - v_a), \end{aligned} \quad (5)$$

where A_{end} is the area of the pipe exit profile section (Figures 4 and 5), v_{end} is the average velocity component normal to the exit plane, and v_a is the advance velocity. According to Equation (5), the determination of thrust depends primarily on the area-averaged velocity at the outlet plane, as all other parameters remain constant for a given geometric configuration.

Typical results for the exit pipe terminations are shown in Figures 21 and 22. Each illustration is divided into three primary scalar and vector fields to characterise the outflow section. The left subfigure shows the velocity component normal to the exit plane (U_z), the central subfigure displays the velocity magnitude ($\text{mag}(U)$), and the right subfigure presents the dynamic pressure profile (p). For a rigorous quantitative assessment, area-

averaged values for each profile and the corresponding exit plane area (A) are provided above their respective subfigures. These averaged quantities are the key parameters required to calculate the net thrust of each configuration across the specified advance velocity range, using the integral momentum relationship defined in Equation (5). This approach enables a definitive evaluation of how each geometric modification influences axial momentum recovery and downstream wake development.

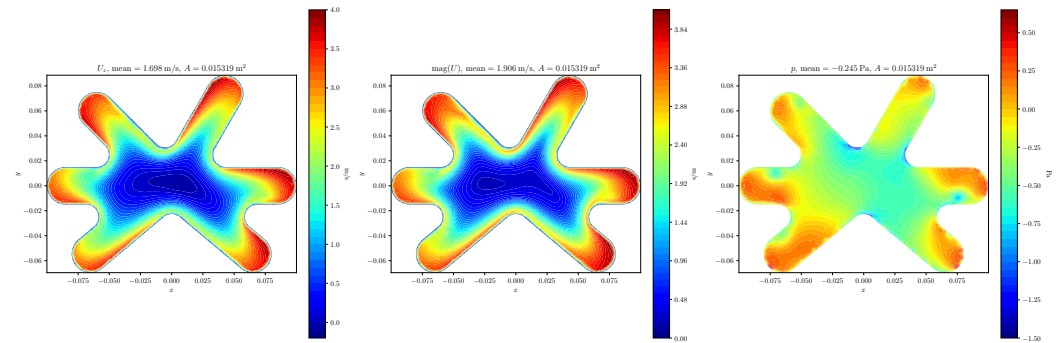


Figure 21. Average result of the outflow section for an advance velocity of 1.0 m/s with a daisy 0° nozzle ((left) U_z —velocity component in the flow direction; (middle) $\text{mag}(U)$ —velocity magnitude; (right) p —dynamic pressure)

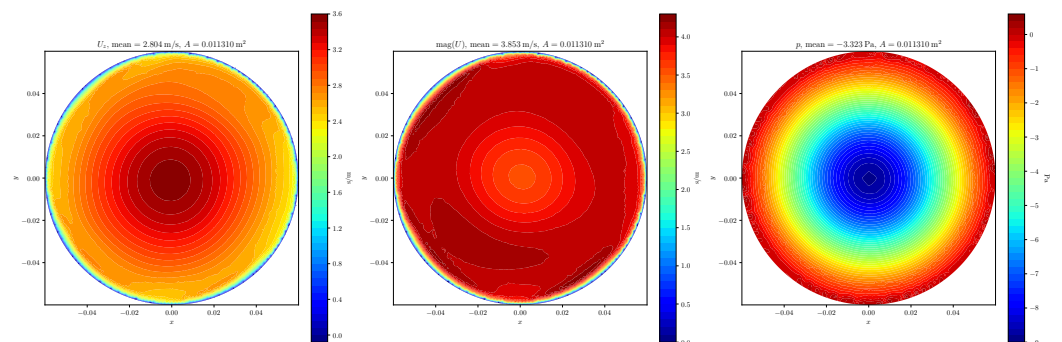


Figure 22. Averaged results at the outflow section for an advance velocity of 1.5 m/s with a 60 mm jet nozzle ((left) U_z —velocity component in the flow direction; (middle) $\text{mag}(U)$ —velocity magnitude; (right) p —dynamic pressure).

3.3.1. Lobed “Daisy” Geometries: Flow Uniformity and Swirl Recovery

The first category of duct terminations featured daisy-shaped, multi-lobed outlets. As shown in the Figure 4, these geometries were designed to manipulate the wake structure by increasing the mixing interface between the high-velocity jet and the ambient fluid. The numerical results for the daisy-shaped nozzle with a multi-lobed outlet rotated by 40° at an advance velocity of 2.0 m/s are presented in Figure 20.

As shown in the left panel of the figure, a pronounced swirl flow was maintained within the internal conduit, generating significant flow instability and preventing the attainment of a steady-state numerical solution. The right panel shows the velocity magnitude in the central YZ plane, where a highly non-stationary flow field is also evident. This intensified swirl and vortex formation are primarily due to the prolonged confinement of the fluid within the pipe, which allows for secondary flow structures to develop and amplify as they progress towards the exit plane. Therefore, the use of a transient numerical framework is essential to accurately capture these unsteady phenomena, as steady-state approximations fail to represent the complex vortical evolution characteristic of such confined propulsion systems.

Representative examples of the time-averaged flow characteristics for the investigated nozzle configurations are shown in Figures 21 and 22. Figure 21 presents the hydrodynamic performance of the 0° daisy nozzle geometry at an advance speed of 1.0 m/s. This configuration serves as the baseline for the lobed nozzle series, highlighting the initial wake development without additional vane-induced swirl.

Comparing the parallel daisy configuration (0°) with the angled variants (20° and 40°), the pressure recovery and velocity distribution showed that the inclined lobed segments effectively counteracted the rotational momentum generated by the ducted rotor. This angular orientation enabled the nozzle to act as a quasi-static stator, recovering kinetic energy from tangential flow components that would otherwise be lost to swirl. However, this flow-straightening effect resulted in a measurable reduction in absolute thrust. At an advance velocity of $V_a = 1.0$ m/s, the parallel daisy nozzle produced approximately 50 % less thrust than the reference cylindrical pipe. Despite this reduction in propulsive force, the exit-plane parameters (Figure 21) confirmed a substantial improvement in flow uniformity and a reduction in swirl intensity compared to the baseline cylindrical duct. This homogenisation of the outlet velocity distribution was identified as a critical mechanism for mitigating flow noise and reducing the vehicle's hydroacoustic signature. Consequently, the daisy-type geometries offered a strategic “stealth” advantage for low-observable underwater platforms, where acoustic discretion was prioritised over the absolute magnitude of axial momentum flux. Furthermore, manipulation of the wake structure facilitated enhanced flow mixing at the jet interface, which is essential for sensitive environmental survey missions.

3.3.2. Convergent Nozzles: Thrust Maximisation Through Jet Acceleration

In contrast to the lobed designs, the convergent nozzles, with exit diameters of 60 mm, 50 mm, and 40 mm, focused on maximising axial momentum flux. The relationship between nozzle contraction and jet enhancement was quantified through velocity field observations, where the 60 mm contraction exhibited the highest exit velocity magnitude.

These observations directly correlated with the integral thrust performance; at an operational speed of 1.0 m/s, the 60 mm convergent nozzle achieved a maximum thrust increase. However, the results indicated that the thruster motor was significantly overloaded when smaller exit diameters were used. The data showed that a substantially more powerful motor would be required to fully exploit the jet acceleration potential of the more aggressive convergent designs, as the current configuration caused excessive rotor loading when the exit area was overly restricted.

3.3.3. Comparative Summary of Performance Trends

Figures 23 and 24 show the thrust, area-averaged velocity magnitude and its component in the z direction, and dynamic pressure at the exit plane as functions of the advanced velocity for each configuration. These datasets allow for a quantitative assessment of the pressure-to-velocity conversion efficiency achieved within the various duct terminations. By correlating the axial momentum flux with the pressure recovery at the exit, it is possible to distinguish the hydrodynamic influence of the daisy-type lobed geometries from the area-contraction effects characteristic of the convergent jet nozzles. This distinction is essential for determining whether thrust gains are primarily due to jet acceleration or to the attenuation of wake turbulence.

The divergent trends between the two geometric families were summarised across the full range of advance velocities. While the convergent nozzles remained the preferred choice for applications requiring high propulsive efficiency and maximum static pull,

the lobed geometries provided a strategic alternative for missions where acoustic discretion was prioritised over pure thrust density.

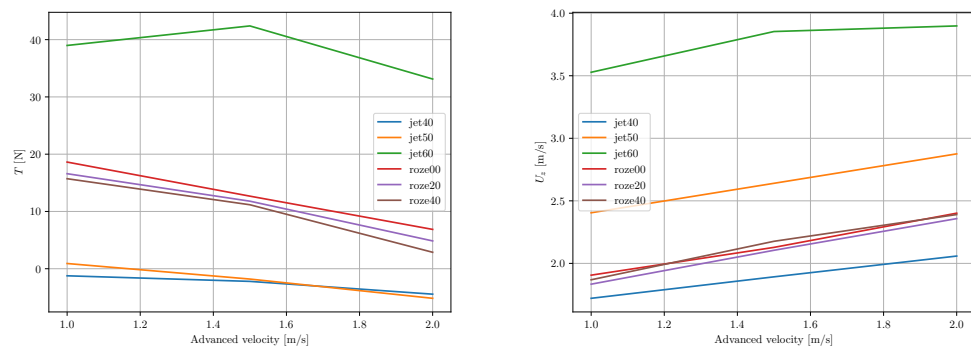


Figure 23. Results for thrust and average velocity component U_z for all nozzle geometries and three different advance velocities.

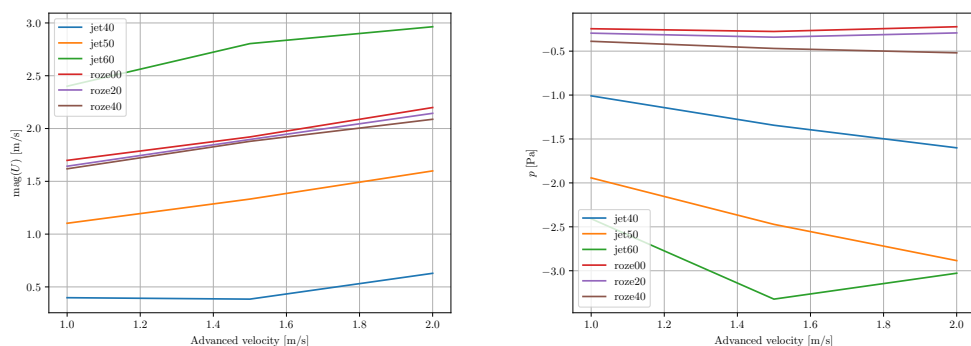


Figure 24. Results for the average velocity magnitude $\text{mag}(U)$ and average pressure p for all nozzle geometries and three different advance velocities.

Furthermore, the robustness of these comparative conclusions was supported by the consistent application of the calibrated actuator disc model across all cases. Although discrepancies in absolute thrust values compared to manufacturer-provided data existed, the relative performance shifts recorded in this investigation were physically meaningful and provided a reliable baseline for selecting the optimal nozzle for the RAS-HA-X25 platform.

4. Discussion

4.1. Hydrodynamic Trade-Offs: Convergent vs. Daisy-Type Geometries

The comparative analysis conducted across the three operational advance velocities (1.0 m/s, 1.5 m/s, and 2.0 m/s) highlights clear hydrodynamic differences between the convergent-jet nozzle configurations and the multi-lobed daisy-type outlet geometries. As shown by the thrust and outlet-plane velocity fields (see Figures 23 and 24), the convergent nozzles primarily enhance performance by accelerating the axial velocity component U_z . This acceleration increases the net momentum flux at the exit, directly in line with the principles of Froude's Momentum Theory [30,31]. However, this improvement is accompanied by an increase in internal dynamic pressure. For the bespoke RAS-HA-X25 AUV, such pressure rises may impose additional structural or energetic constraints that must be considered during the integration phase.

In contrast, daisy-type lobed geometries induce a more complex redistribution of momentum. Although the increased wetted perimeter introduces additional viscous losses, the lobes promote enhanced flow mixing and partially suppress the concentrated swirl-driven wake typically observed behind high-solidity ducted rotors [4]. This behaviour

aligns with mechanisms exploited in pump-jet and stator-assisted propulsion systems documented by Shirazi [24] and Xia [28]. While these geometries do not outperform convergent nozzles in absolute thrust generation, they produce a more uniform exit-plane velocity distribution. Such homogenisation is a critical advantage for hydroacoustic reduction and flow-noise mitigation—an increasingly important factor in the design of low-observable underwater platforms and sensitive environmental survey AUVs.

4.2. Numerical Methodology and Simulation Fidelity

The use of a transient simulation framework was essential for accurately capturing the unsteady internal flow structures in pipe-embedded thrusters. Steady-state RANS solvers could not achieve convergence because of strong recirculation zones, vortical structures, and intermittent separation phenomena within the extended duct. Such unsteadiness is typical of confined propulsion systems where the duct length permits the amplification of secondary flow structures [24,25]. Each transient simulation was initiated with a pre-conditioned steady-state solution, and temporal averaging was performed over the final 0.2 s, so the resulting thrust values represent a statistically stable operating condition. This approach filters out high-frequency oscillations that do not contribute to net propulsive performance while preserving the physical integrity of the mean flow.

4.3. Actuator Disc Calibration and Discrepancies

A discrepancy of approximately 15 % was observed between the thrust predicted using the resolved rotor model ($T_p = 48 \text{ N}$ at $V_a = 1 \text{ m/s}$) and the actuator disc model ($T_d = 57.22 \text{ N}$). This difference is attributed primarily to the absence of blade-tip losses, hub-vortex effects, and spanwise loading variations in the actuator disc approximation—phenomena inherently captured in resolved-rotor CFD but simplified in the pressure-jump approach [35].

4.4. Discrepancy Between Calculated Thrust and Manufacturer Data

The numerical analysis of thrust performance revealed a significant divergence between the manufacturer's empirical data and the results of the present simulation. Specifically, for the bollard pull condition at 3000 RPM, the manufacturer reports a thrust force of approximately 160 N. In contrast, the current CFD model yielded a value of approximately 50 N. This substantial discrepancy necessitates a rigorous evaluation of the underlying simulation parameters and geometric assumptions.

The simulation was designed to replicate the experimental conditions of a standard bollard pull test ($V_\infty = 0$). To ensure computational accuracy, turbulent wall functions were used as boundary conditions, which are highly effective for y^+ values above 30; this condition was strictly maintained throughout the study (Figure 6). To approximate the static thrust, an extrapolation method was used, projecting data from higher advance ratios down to zero velocity.

Several factors may account for this variance:

- **Geometric Fidelity:** It is highly probable that the publicly available 3D CAD models of the T500 thruster deviate from the actual production hardware. Such geometric inconsistencies often lead to marked differences in hydrodynamic performance.
- **Blockage Effects:** The presence of internal conduits or mounting structures in the simulation may introduce blockage effects that are absent in the manufacturer's open-water testing environment.

Despite the magnitude of this absolute difference, the performance trends, characterised by the slope of the characteristic curves (Figure 13), are likely consistent with the simulated results. Consequently, the actuator disc model remains a robust and suitable

framework for comparative nozzle studies. As each nozzle configuration was evaluated using the same momentum source, the relative performance gains remain physically meaningful. This ensures that the selection of the optimal geometry for the Autonomous Underwater Vehicle's (AUV) operational advance ratio ($J = 0.167$) is based on a reliable comparative metric.

5. Conclusions

This study demonstrated that the hydrodynamic performance of the T500 thruster embedded in an AUV duct is highly sensitive to duct-exit morphology, and that the optimal outlet geometry depends directly on the intended mission profile. The comparative analysis of six outlet configurations showed a clear trade-off between thrust generation and wake conditioning. In particular, the convergent jet nozzles were found to favour axial momentum recovery and thrust production, whereas the daisy shaped lobed terminations were more effective at homogenising the outflow and reducing swirl intensity.

Among the convergent designs, the 60 mm nozzle provided the best overall propulsion performance under the tested operating conditions. It achieved a peak thrust of approximately 42 N at an advance velocity of 1.5 m/s, indicating that moderate contraction can enhance exit plane jet acceleration without imposing excessive hydraulic penalties on the propulsion system. By contrast, the more aggressive 50 mm and 40 mm convergent nozzles increased back pressure sufficiently to overload the motor and degrade net performance, in some cases leading to suboptimal or even negative thrust within the available power range. These results show that stronger geometric contraction does not automatically translate into better propulsive performance; its benefit is constrained by the motor/duct coupling and the allowable operating envelope of the thruster. For AUVs intended for high speed transit or extended range missions, the 60 mm convergent nozzle therefore represents the most suitable engineering solution among the tested cases.

The daisy shaped lobed nozzles showed a different but equally important hydrodynamic behaviour. Although they generated substantially lower absolute thrust than the convergent geometries, with the 0° variant producing about 18 N at 1.0 m/s, roughly 50% lower than the reference cylindrical duct, they improved flow uniformity and suppressed concentrated swirl in the wake. The angled lobed variants, especially the 20° and 40° cases, acted in practice as quasi static stators by recovering part of the kinetic energy associated with rotor induced swirl and stabilising the downstream flow. This indicates that lobed exits are not optimal when maximum thrust density is the primary objective, but they offer a meaningful advantage for missions where wake control, reduced flow non-uniformity, and lower hydroacoustic detectability are more important than absolute speed. Their main implication is therefore strategic rather than purely propulsive: they may support minimally observable or acoustically discreet AUV operations.

From a modelling perspective, the results also showed that the internal duct flow is strongly unsteady and involves secondary flow development, vortex evolution, and non trivial axial momentum redistribution. As a result, steady state methods are not sufficient for capturing the relevant propulsion physics. Transient CFD was necessary to resolve the relative performance differences between outlet geometries with acceptable fidelity. At the same time, the calibrated actuator disk approach proved to be a computationally efficient and reliable method for comparative design assessment. Although discrepancies remained between predicted absolute thrust values and manufacturer reported data, the relative trends between geometries were robust because all cases were evaluated using the same calibrated framework. The main implication is that actuator-disk-based transient CFD can serve as an effective intermediate methodology for nozzle comparison and prelimi-

nary optimisation, bridging the gap between low order propulsion models and full rotor resolved simulations.

Overall, the study establishes that pipe exit morphology should be treated as a mission dependent design variable in embedded AUV propulsion. Convergent outlets, particularly the 60 mm configuration, are preferable when maximising axial thrust, cruising efficiency, and range is the dominant requirement. Daisy type lobed outlets are preferable when flow conditioning and potential hydroacoustic signature reduction take priority. This provides a practical framework for multi objective optimisation of embedded propulsion modules in next generation autonomous underwater vehicles.

Future work will focus on experimentally validating these numerical trends in a towing tank using the RAS-HA-X25 test platform. Such measurements will allow for a rigorous assessment of actuator disc fidelity and a definitive quantification of the discrepancy between numerical predictions and manufacturer-reported values. Additionally, coupling the present CFD framework with an aeroacoustic solver will clarify the role of lobed geometries in noise reduction, enabling multi-objective optimisation of embedded propulsion modules for next-generation autonomous and hybrid underwater vehicles.

Author Contributions: Conceptualization, A.G., M.P., R.K., D.M. and A.A.; Methodology, A.G., M.D. and A.A.; Software, A.G., M.D. and D.M.; Validation, A.G.; Formal analysis, A.G., M.D. and D.M.; Investigation, A.G., M.D. and D.M.; Resources, A.G. and A.A.; Data curation, A.G. and M.D.; Writing—original draft, A.G., M.D. and D.M.; Writing—review & editing, A.G., M.P., R.K. and A.A.; Visualization, A.G., M.P. and R.K.; Supervision, A.G., M.P., R.K. and A.A.; Project administration, A.A.; Funding acquisition, R.K. and A.A. All authors have read and agreed to the published version of the manuscript.

Funding: This research was partially funded by the Ministry of Defence of the Republic of Slovenia and carried out under the auspices of the European Defence Agency (EDA) within the framework of the Swarm of Biomimetic Underwater Vehicles II (SABUVIS II) project (EDA Category B Project No. B-746-ESM1-G). The consortium, comprising partners from Poland, Germany, Slovenia, and Portugal, collaboratively developed and evaluated three distinct swarm concepts: the Low-Cost AUV Swarm (LAUVS); the Biomimetic Littoral Intelligent Swarm Technology for Underwater Navigation and Autonomous Vessel Intelligence (BIO LIST-U NAVI); and a hybrid swarm concept integrating LAUVs with Autonomous Surface Vehicles (ASVs) (LAUV-ASV-S). This research was also partially funded by the Slovenian Research and Innovation Agency (Grants No. L7-60164 and P2-0228).

Data Availability Statement: The original contributions presented in this study are included in the article. Further inquiries can be directed to the corresponding author(s).

Conflicts of Interest: The authors declare no conflicts of interest.

References

- Oweis, G.F.; Ceccio, S.L. Instantaneous and time-averaged flow fields of multiple vortices in the tip region of a ducted propulsor. *Exp. Fluids* **2005**, *38*, 615–636. [[CrossRef](#)]
- Wadoo, S.; Kachroo, P. *Autonomous Underwater Vehicles: Modeling, Control, and Simulation*; CRC Press: Boca Raton, FL, USA, 2011. [[CrossRef](#)]
- Yan, Z.; Wang, Y.; Pan, C. *Autonomous Underwater Vehicles: Localization, Tracking, and Formation*; Springer Nature: Berlin/Heidelberg, Germany, 2021.
- Carlton, J. *Marine Propellers and Propulsion*; Butterworth-Heinemann: Oxford, UK, 2018.
- De Barros, E.A.; Freire, L.O.; Dantas, J.L. Development of the Pirajuba AUV. In *IFAC Proceedings Volumes*; Elsevier: Amsterdam, The Netherlands, 2010; Volume 43, pp. 102–107. [[CrossRef](#)]
- Palmer, M.R. Analysis of Propulsion and Manoeuvring Characteristics of Survey-Style AUVs. Ph.D. Thesis, University of Southampton, Southampton, UK, 2009.
- Quang, T.N. Detailed Design of a Thruster Solution for Mass-Market ROVs. Master's Thesis, University of Stavanger, Stavanger, Norway, 2016.

8. Pereira, P.; Silva, R.; Marques, J.; Campilho, R.; Matos, A.; Pinto, A. Raya: A Bio-Inspired AUV for Inspection and Intervention of Underwater Structures. *IEEE Access* **2025**, *13*, 71253–71274. [[CrossRef](#)]
9. International Towing Tank Conference. *Recommended Procedures and Guidelines 7.5-02-03-02.1: Open Water Test*; Revision 04; International Towing Tank Conference: Edinburgh, UK, 2021.
10. Bhattacharyya, A.; Krasilnikov, V.; Steen, S. A CFD-based scaling approach for ducted propellers. *Ocean Eng.* **2016**, *123*, 116–130. [[CrossRef](#)]
11. Baltazar, J.M.; Rijpkema, D.; Falcão de Campos, J.; Bosschers, J. Prediction of the open-water performance of ducted propellers with a panel method. *J. Mar. Sci. Eng.* **2018**, *6*, 27. [[CrossRef](#)]
12. Li, H.; Jin, Z.; Huang, J.; Liu, Z.; Peng, X.; Wang, Z.; Chen, Y. Hybrid-Driven Disc-Shaped Autonomous Underwater Vehicle with High Maneuverability and Gliding Capability: Design and Experiments. *IEEE Robot. Autom. Lett.* **2025**, *11*, 946–953. [[CrossRef](#)]
13. Suzuki, H.; Sakaguchi, J.; Inoue, T.; Watanabe, Y.; Yoshida, H. Evaluation of methods to estimate hydrodynamic force coefficients of underwater vehicle based on CFD. In *IFAC Proceedings Volumes*; Elsevier: Amsterdam, The Netherlands, 2013; Volume 46, pp. 197–202. [[CrossRef](#)]
14. Joung, T.H.; Choi, H.S.; Jung, S.K.; Sammut, K.; He, F. Verification of CFD analysis methods for predicting the drag force and thrust power of an underwater disk robot. *Int. J. Nav. Archit. Ocean Eng.* **2014**, *6*, 269–281. [[CrossRef](#)]
15. Allotta, B.; Costanzi, R.; Ridolfi, A.; Salvetti, O.; Reggiannini, M.; Kruusmaa, M.; Salumée, T.; Lane, D.; Frost, G.; Tsiogkas, N.; et al. Identification of hydrodynamic parameters of the Typhoon AUV. *J. Mar. Sci. Eng.* **2018**, *6*, 75. [[CrossRef](#)]
16. da Silva Costa, G.; Ruiz, A.; Reis, M.; da Cunha Lima, A.; de Almeida, M.; da Cunha Lima, I. Numerical analysis of stability and manoeuvrability of Autonomous Underwater Vehicles (AUV) with fishtail shape. *Ocean Eng.* **2017**, *144*, 320–326. [[CrossRef](#)]
17. Huang, H.; Bian, X.; Jiang, T.; Tang, Q.; Qin, H.; Wang, Z. Hydrodynamic analysis and manipulation control on a streamlined I-AUV. *ISA Trans.* **2024**, *153*, 453–466. [[CrossRef](#)]
18. Bao, C.; Xu, H.; Wu, H. A review of underwater vehicle motion stability. *Ocean Eng.* **2023**, *287*, 115735. [[CrossRef](#)]
19. Wang, H.; Xiang, X.; Xiang, G.; Liu, C.; Yang, L. An improved body force method for simulation of self-propulsion AUV with ducted propeller. *Ocean Eng.* **2023**, *281*, 114731. [[CrossRef](#)]
20. Cai, B.; Mao, X.; Xu, Q.; Chai, W.; Tian, B.; Qiu, L. Simulation of the interaction between ship and ducted propeller with a modified body force method. *Ocean Eng.* **2022**, *249*, 110950. [[CrossRef](#)]
21. Einarsson, G.S. Model Predictive Control for the BlueROV2. Master's Thesis, Norwegian University of Science and Technology, Trondheim, Norway, 2020.
22. von Benzön, M.; Sørensen, F.F.; Uth, E.; Jouffroy, J.; Liniger, J.; Pedersen, S. An open-source benchmark simulator: Control of a bluerov2 underwater robot. *J. Mar. Sci. Eng.* **2022**, *10*, 1898. [[CrossRef](#)]
23. Riccaro, L. Modelling of ROVs: Accuracy Assessment of ODE-Based Simulators. Master's Thesis, Politecnico di Milano, Milan, Italy, 2023.
24. Shirazi, S.A.; Nazari, M.; Manshadi, M.D. Fluid flow investigation of a full-scale pump-jet thruster. *J. Mar. Sci. Appl.* **2019**, *18*, 167–177. [[CrossRef](#)]
25. Zhang, Q.; Jaiman, R.K. Numerical analysis on wake dynamics of a ducted propeller. *Ocean Eng.* **2019**, *171*, 202–224. [[CrossRef](#)]
26. Xia, H.G.; Wang, P.; Jin, Z.; An, X.; Ding, Y. Maneuverability analysis of thrust vectoring ducted propeller with deflector. *Ocean Eng.* **2020**, *213*, 107614. [[CrossRef](#)]
27. Xia, H.; Wang, P.; Dong, H.; Jin, Z.; An, X.; Fang, L. Hydrodynamic characteristics of a new thrust vectoring ducted propeller with slotted nozzle. *Ocean Eng.* **2022**, *266*, 112805. [[CrossRef](#)]
28. Xia, T.; Liu, S.; Wang, T.; Li, J.; Lin, W.; Zhang, B.; Cai, Y.; Xu, W. Experimental and numerical analysis on two-way hull-propeller coupled effect of a fully-actuated AUV. *Ocean Eng.* **2025**, *321*, 120340. [[CrossRef](#)]
29. Du, X.; Cui, H.; Zhang, Z. Dynamics model and maneuverability of a novel AUV with a deflectable duct propeller. *Ocean Eng.* **2018**, *163*, 191–206. [[CrossRef](#)]
30. Halliday, D.; Resnick, R.; Walker, J. *Principles of Physics: Extended, International Adaptation*, 12th ed.; John Wiley & Sons: Hoboken, NJ, USA, 2023. [[CrossRef](#)]
31. Houghton, E.L.; Carpenter, P.W. *Aerodynamics for Engineering Students*, 8th ed.; Elsevier: Amsterdam, The Netherlands, 2024.
32. Jiang, Y.; Feng, C.; He, B. AUV thruster fault diagnosis based on comprehensive model. *IEEE Trans. Reliab.* **2026**, 1–45. [[CrossRef](#)]
33. Liu, J.; Wang, M.; Yu, F.; Gao, S.; Yan, T.; He, B. Numerical study on the hull-propeller interaction of autonomous underwater vehicle. *Ocean Eng.* **2023**, *271*, 113777. [[CrossRef](#)]
34. Hong, L.; Wang, X.; Zhang, D.S. CFD-based hydrodynamic performance investigation of autonomous underwater vehicles: A survey. *Ocean Eng.* **2024**, *305*, 117911. [[CrossRef](#)]
35. Sørensen, J.N. Aerodynamic aspects of wind energy conversion. *Annu. Rev. Fluid Mech.* **2011**, *43*, 427–448. [[CrossRef](#)]
36. Vega, E.P.; Chocron, O.; Benbouzid, M. A flat design and a validated model for an AUV reconfigurable magnetic coupling thruster. *IEEE/ASME Trans. Mechatron.* **2016**, *21*, 2892–2901. [[CrossRef](#)]

37. Fagundes Gasparoto, H.; Chocron, O.; Benbouzid, M.; Siqueira Meirelles, P.; Saraiva Ferreira, L.O. Torque analysis of a flat reconfigurable magnetic coupling thruster for marine renewable energy systems maintenance AUVs. *Energies* **2018**, *12*, 56. [[CrossRef](#)]
38. Liu, T.; Hu, Y.; Xu, H.; Wang, Q.; Du, W. A novel vectored thruster based on 3-RPS parallel manipulator for autonomous underwater vehicles. *Mech. Mach. Theory* **2019**, *133*, 646–672. [[CrossRef](#)]
39. Liu, T.; Hu, Y.; Xu, H.; Zhang, Z.; Li, H. Investigation of the vectored thruster AUVs based on 3SPS-S parallel manipulator. *Appl. Ocean Res.* **2019**, *85*, 151–161. [[CrossRef](#)]
40. Geng, B.; Hu, Z.; Lin, Y. Hydrodynamic characteristics of synthetic jet-steered underwater vehicle. *Ocean Eng.* **2018**, *70*, 1–13. [[CrossRef](#)]
41. Hai, H.; Zexing, Z.; Jiyong, L.; Qirong, T.; Wanli, Z.; Wang, G. Investigation on the mechanical design and manipulation hydrodynamics for a small sized, single body and streamlined I-AUV. *Ocean Eng.* **2019**, *186*, 106106. [[CrossRef](#)]
42. Huang, K.; Li, W.; Cao, S.; Xiao, P.; Wu, X.; Fang, H. Fault diagnosis in autonomous underwater vehicle propulsion system: A systematic review. *Ships Offshore Struct.* **2025**, 1–29. [[CrossRef](#)]
43. Lyu, B.; Dowling, A.P. An experimental study of the effects of lobed nozzles on installed jet noise. *Exp. Fluids* **2019**, *60*, 176. [[CrossRef](#)]
44. Cheng, Z.; Smoker, B.; Kashyap, S.; Burella, G.; Jaiman, R.K. Cavitating wake dynamics and hydroacoustics performance of marine propeller with a nozzle. *Phys. Fluids* **2025**, *37*, 015128. [[CrossRef](#)]
45. Chen, M.; Liu, J.; Si, Q.; Liang, Y.; Jin, Z.; Yuan, J. Investigation into the hydrodynamic noise characteristics of electric ducted propeller. *J. Mar. Sci. Eng.* **2022**, *10*, 378. [[CrossRef](#)]
46. Celik, I.B.; Ghia, U.; Roache, P.J.; Freitas, C.J.; Coleman, H.; Raad, P.E. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *J. Fluids Eng.* **2008**, *130*, 078001. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.