

Article

Automated Design, Evaluation, and Optimization of 2D Rotor Blade Sections for Tidal Stream Turbines Using HEEDS

Soonhyun Lee ¹, Hyungju Kim ^{1,*}  and Sooyeon Kwon ²¹ Department of Mechanical and Industrial Engineering, Norwegian University of Science and Technology, 7034 Trondheim, Norway; soonhyun.lee@ntnu.no² Department of Fishing Vessel Safety Research, Korea Maritime Transportation Safety Authority, Sejong-si 30100, Republic of Korea

* Correspondence: hyung-ju.kim@ntnu.no

Abstract

An automated CFD-based workflow for the design, evaluation, and comparative optimization of 2D tidal-stream turbine blade sections is presented for early-stage design exploration. The workflow is intended to efficiently derive an improved section using a consistent and higher fidelity evaluation approach, which is particularly relevant for floating tidal concepts where the effective angle of attack can vary. HEEDS is used to manage a SHERPA optimization loop, while candidate geometries are regenerated in Rhino Grasshopper through a control point parameterization with thickness bounds and smooth interpolation. STAR-CCM+ simulations are executed in an automated manner and the resulting lift and drag responses are returned to HEEDS to evaluate performance over four representative angles of attack, 0, 3, 6, and 9 deg. A total of 1000 design evaluations are conducted for a baseline NACA 63–815 section at Reynolds number 1×10^7 , using a two metric formulation that targets high mean lift to drag ratio while limiting the maximum drag coefficient within the same angle set. The optimization history shows rapid early improvement followed by a plateau and identifies a final best design at Design 746. Compared with the original section, the optimized section increases lift and improves the lift-to-drag ratio across the operating range, while keeping the peak drag constrained. Cavitation inception characteristics also improve, with the optimized section delaying inception at the same lift criterion and sustaining a cavitation free state at higher lift for the same cavitation number. Pressure coefficient distributions indicate that these changes are primarily associated with altered suction side loading in the front to mid chord region and modified pressure recovery behavior. A preliminary full 3D RANS CFD rotor comparison under a prescribed rotor geometry further shows that the optimized section can improve rotor power performance in the main operating TSR range, although the benefit becomes limited at high TSR.

Academic Editors: Alberto Peña
and Constantine Michailides

Received: 27 May 2026

Revised: 20 June 2026

Accepted: 22 June 2026

Published: 24 June 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:** automated design; unsteady reynolds-averaged navier-stokes; computational fluid dynamics; HEEDS; Rhino Grasshopper; tidal-stream turbine; NACA63–815; 2D section

1. Introduction

The use of renewable energy is increasing, and there is growing interest in low-carbon energy sources that can provide more predictable output than weather-driven resources. Marine renewable energy is one option in this context, especially at sites where the resource is strong and steady. Tidal-stream energy is of particular interest because tidal currents

are predictable and repeatable over time, and tidal-stream turbines can generate electricity by converting the kinetic energy of these currents using rotor-based devices in energetic coastal channels.

The technical potential of tidal energy, including both tidal range and tidal current resources, has been discussed for many years. Charlier (2003) [1] noted that tidal current power can be highly predictable, with annual variability below 5 percent, and that tidal installations can have long service lives, while deployment has remained limited due to high upfront cost, environmental concerns, and delays in execution. Similar barriers are still observed in practice, and large-scale deployment remains limited. At the same time, academic interest has remained active. Khojasteh et al. (2023) [2] conducted a large-scale bibliometric review of more than 8000 wave and tidal energy papers published from 2003 to 2021, identifying flow dynamics, tidal turbines, and turbine design as a major research theme and noting a recent shift toward performance and optimization topics. More recently, Lee and Kim (2025) [3] performed a Scopus-based bibliometric analysis of 791 tidal energy publications, classified the literature into four main clusters including tidal turbine design, turbulence analysis, resource assessment, and feasibility, and discussed future research directions for advancing tidal energy technologies.

Tidal stream turbines have historically been developed mainly in horizontal-axis and vertical-axis configurations, and many early large-scale systems adopted fixed-bottom installation. Fixed-bottom concepts are typically supported by piles or gravity-based foundations on the seabed, which can provide stiff support condition and simplify station keeping, but their applicability is often constrained by water depth, seabed conditions, installation and permitting requirements [4]. In recent years, a wider range of concepts has been proposed, including kite-type systems [5] and floating platforms, to reduce dependence on complex seabed foundations and to improve access for installation and maintenance. Floating tidal platforms are particularly attractive because they can broaden feasible sites and support retrieval-based maintenance, although they introduce platform motion and coupled effects that can influence rotor loading and performance. Representative floating platform developments include PLAT-I [6], Orbital Marine Power's O2 [7], and the ATIR platform concept [8]. Motivated by this direction, the aim of this study is to develop an optimized turbine suitable for floating tidal platforms, and as a foundational step this study focused on 2D blade-section optimization using an automated design and evaluation process.

Most tidal turbine design studies start with 2D blade-section polars and then use rotor-level models to translate section performance into turbine-scale power and loads. Earlier work largely relied on BEM (Blade element method) or BEMT (Blade element momentum theory) formulations coupled with low-order polar prediction tools, while recent studies increasingly incorporate CFD (Computational fluid dynamics) and experimental evidence to improve fidelity and reduce uncertainty in performance assessment. Accordingly, a substantial body of research has used 2D section analyses as a foundation for blade and rotor design by combining XFOIL or RFOIL to generate the polar data with reasonable validation and rotor synthesis models. Batten et al. (2006) [9], for example, assessed NACA 63-series sections using XFOIL together with cavitation-tunnel measurements and propagated the section characteristics to rotor performance using a BEM, while Grasso (2012) [10] optimized tidal turbine airfoils under cavitation and geometric constraints using RFOIL. At the rotor scale, Goundar and Ahmed (2013) [11] proposed an HF10XX hydrofoil series tailored to different radial positions and designed a 10 m-class turbine using optimized chord and twist distributions, supported by section verification through experiments and CFD. Jo et al. (2017) [12] similarly screened candidate sections using 2D CFD and designed and validated a 10 m turbine using BEMT and 3D CFD with towing-

tank testing. In parallel, Morgado et al. (2016) [13] showed that transition modeling can materially affect low-Reynolds-number polar reliability when comparing XFOil and RANS (Reynolds-Averaged Navier-Stokes) CFD, and Vogel et al. (2018) [14] demonstrated that blockage-corrected BEM formulations can be necessary in confined-flow applications due to their impact on both power and thrust predictions.

Recent studies increasingly adopt automated, computation-based design workflows, and an emerging trend is to incorporate machine-learning-based surrogates to reduce the cost of repeated performance evaluations in optimization loops. Zhu et al. (2020) [15] implemented a BEM-based design loop with CFD validation, parameterizing chords using a Bezier curve and applying an RBF (Radial basis function) surrogate together with NSGA-II multi-objective optimization to refine chords and pitch distributions. At the section level, Lan et al. (2021) [16] optimized a baseline NACA4415 airfoil using a genetic algorithm with a weighted single-objective formulation, where the objective emphasized lift-to-drag improvement under lift and geometry constraints and the optimized section was evaluated using CFD. Similarly, section-level optimization has also been reported for NACA 6-series hydrofoils widely used in tidal applications, including NACA 63–815, also denoted as NACA 63815. el Sheshtawy et al. (2021) [17] parameterized NACA 63815 using Bezier curves and applied NSGA-II to improve glide ratio while accounting for cavitation-related behavior through pressure-based metrics, and they adopted XFOil for large-scale screening after benchmarking both XFOil and RANS CFD against experiments, which enabled evaluation of about 2480 candidate shapes within a practical runtime. Zhang et al. (2023) [18] also used NACA63815 as a baseline in an automated optimization loop that couples CST (Class-shape transformation) parameterization, XFOil evaluation, and a multi-island genetic algorithm within iSight, with representative designs checked by CFD. For the NACA63815 case, they reported a 15 percent increase in lift coefficient and a 16 percent increase in lift-to-drag ratio at an angle of attack of 5 deg under drag and thickness constraints. Furthermore, Yeo et al. (2022) [19] jointly optimized spanwise NACA profile selection and local chord and twist using an NSGA-based approach coupled with an improved BEM model and XFOil-generated polars and validated the BEM predictions against experimental power and thrust data while presenting Pareto-optimal designs that increased power coefficient and reduced thrust and flap-wise bending moment. More recently, automation has expanded to surrogate and learning-based models to accelerate evaluation in coupled problems: Wang et al. (2023) [20] proposed a CNN (Convolutional neural network) -FEM (Finite element method) static fluid-structure interaction framework for flexible hydrofoils, Li et al. (2024) [21] coupled a deep-learning surrogate for 2D hydrofoil polars with a BEMT rotor model and applied NSGA-II optimization with representative designs confirmed by CFD, and Xu et al. (2024) [22] introduced a coupled CNN-BEM and FEM workflow for multi-objective optimization of a composite turbine blade. Overall, these efforts illustrate a shift from manual trial-and-error toward automated and optimization-driven procedure, motivating a systematic 2D blade-section design and evaluation process as a practical foundation for reproducible early-stage rotor development.

In summary, previous studies demonstrate a clear shift from manual trial-and-error toward automated optimization workflows, often supported by reduced-order tools and surrogate models to lower the cost of repeated evaluations. However, a practical and repeatable section-level design procedure based on consistent CFD assessment remains relatively limited, particularly for comparative evaluation under varying operating conditions. In this context, this paper presents an automated HEEDS-based workflow for CFD-driven design and evaluation of 2D tidal-stream turbine blade sections. Compared with previous studies that primarily relied on XFOil, BEM-based evaluation, the present study applies a consistent CFD-based assessment throughout the automated design procedure. The proposed

procedure evaluates multiple representative angles of attack and examines hydrodynamic performance together with cavitation inception tendency under identical numerical settings. This provides a reproducible basis for comparative section-level design assessment of tidal-stream turbine blades. This structure also makes the procedure adaptable to different operating conditions, objective functions, and geometric constraints with limited manual modification. The automation is achieved by integrating HEEDS with Rhino Grasshopper for parametric geometry regeneration and with STAR-CCM+ for scripted CFD execution and response extraction, enabling a closed-loop evaluation of candidate sections under identical numerical settings. Using the resulting CFD database, angle-of-attack-dependent trends in lift, drag, and lift-to-drag ratio are analyzed, and the corresponding changes in minimum pressure, cavitation inception tendency, and surface pressure distributions are examined to explain the physical mechanisms behind the observed trends. In addition, a preliminary full 3D RANS CFD rotor comparison is carried out by applying the optimized section to a prescribed tidal turbine blade geometry in order to examine whether the section-level improvement can be reflected in rotor-level performance tendency. Finally, the hydrodynamic performance of the existing and optimized sections is quantitatively compared under the same conditions, and the implications of the optimized section are further discussed from the perspective of 3D rotor performance, providing a basis for the conclusions drawn from the trend and hydrodynamic analysis.

2. Methodology

This study investigates a 2D blade-section optimization process as a foundational step toward 3D tidal-stream turbine design. The overall optimization procedure is summarized in Figure 1. First, the design parameters are defined in terms of 2D section control-point coordinates. The candidate geometry is then generated automatically using Rhino/Grasshopper and exported as an IGS file for analysis. Each candidate is evaluated through CFD simulations, which are executed in an automated manner via JavaScript. Based on the CFD outputs, objective and constraint values are computed during the performance evaluation stage. Finally, HEEDS iteratively updates the design parameters using a SHERPA (Simultaneous hybrid exploration that is robust, progressive and adaptive) based optimization loop and selects the best-performing candidate as the optimized geometry.

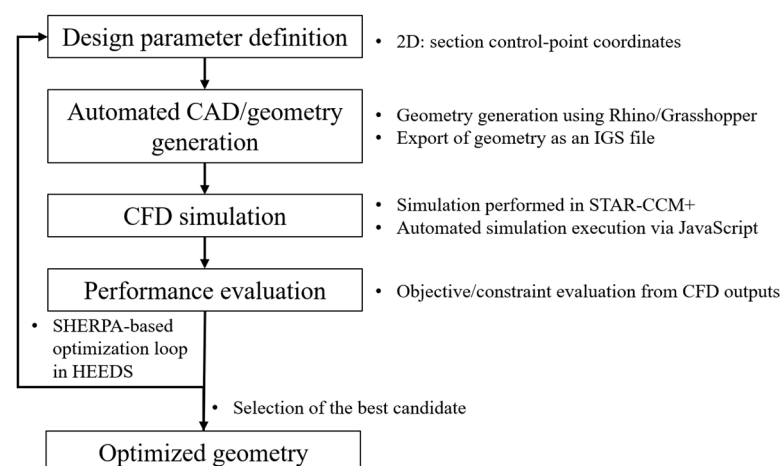


Figure 1. Automated HEEDS based workflow for 2D blade section optimization.

In terms of the SHERPA, it is a hybrid and adaptive search strategy that runs multiple search methods simultaneously rather than sequentially, combining global exploration with local improvement. During the search, it exploits the strengths of each method and automatically reduces the contribution of methods that become ineffective, while adjusting

method-specific tuning parameters based on the information accumulated from completed evaluations. This progressive learning behavior allows SHERPA to adapt to the observed characteristics of the design space without requiring manual algorithm tuning, so the optimization can be specified primarily through the design variables and the available evaluation budget. In the present workflow, each evaluation corresponds to an automated CFD run for a candidate geometry generated from section control-point coordinates, and SHERPA iteratively proposes new candidates to improve the objective while satisfying the imposed constraints.

2.1. Design Parameter Definition

As a foundational step for tidal-stream turbine blade design, a two-dimensional (2D) section optimization problem is formulated using the NACA 63–815 profile as the baseline geometry. The baseline section is first imported into the CAD (Computer-aided design) environment and resampled into a set of discrete coordinates to ensure a consistent geometric representation throughout the automated design loop. In this study, 100 points are extracted from the reference NACA 63–815 section and used as the initial description of the upper (suction) and lower (pressure) surfaces.

To enable controlled shape modification within the proposed workflow, a control-point based parameterization is implemented in Rhino/Grasshopper. The chordwise locations of the control points are fixed, and the design variables are defined as the surface ordinates at selected chordwise positions. As shown in Figure 2, five control points are assigned on each side of the section at normalized chord positions $x/c = 0.05, 0.25, 0.40, 0.60$, and 0.80 . These locations are selected to reflect distinct hydrodynamic roles along the chord. The forward point at $x/c = 0.05$ influences the leading-edge curvature and the initial suction peak. The mid-front points at $x/c = 0.25$ and 0.40 shape the primary camber and thickness development where much of the lift is generated. The mid-aft point at $x/c = 0.60$ affects pressure recovery and separation tendency. The aft point at $x/c = 0.80$ adjusts the trailing region shape that influences wake formation and profile drag while avoiding direct manipulation of the trailing edge. This placement provides sufficient flexibility to modify the camber and thickness distribution over the main load-carrying region while maintaining geometric stability near the leading edge and trailing edge. For each candidate design generated during optimization, the section curves are reconstructed in Grasshopper by smoothly interpolating the control points on the upper and lower surfaces while keeping the leading-edge and trailing-edge connectivity consistent with the baseline.

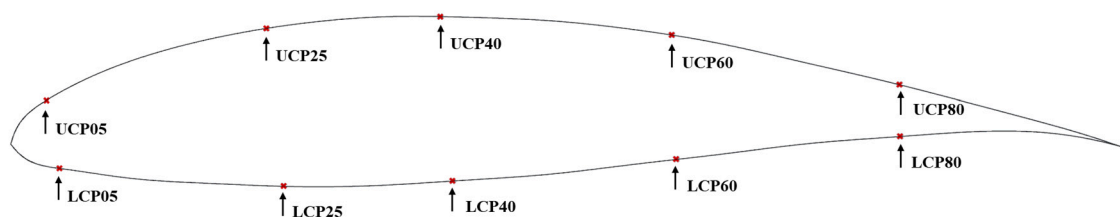


Figure 2. Control points of the NACA 63–815 section.

2.2. Automated Geometry Generation

For automated geometry generation, Figure 3 illustrates the Grasshopper definition used to regenerate a candidate 2D section and export it for CFD evaluation, and the workflow is executed sequentially within each optimization iteration as Process 1, Process 2, and Process 3.

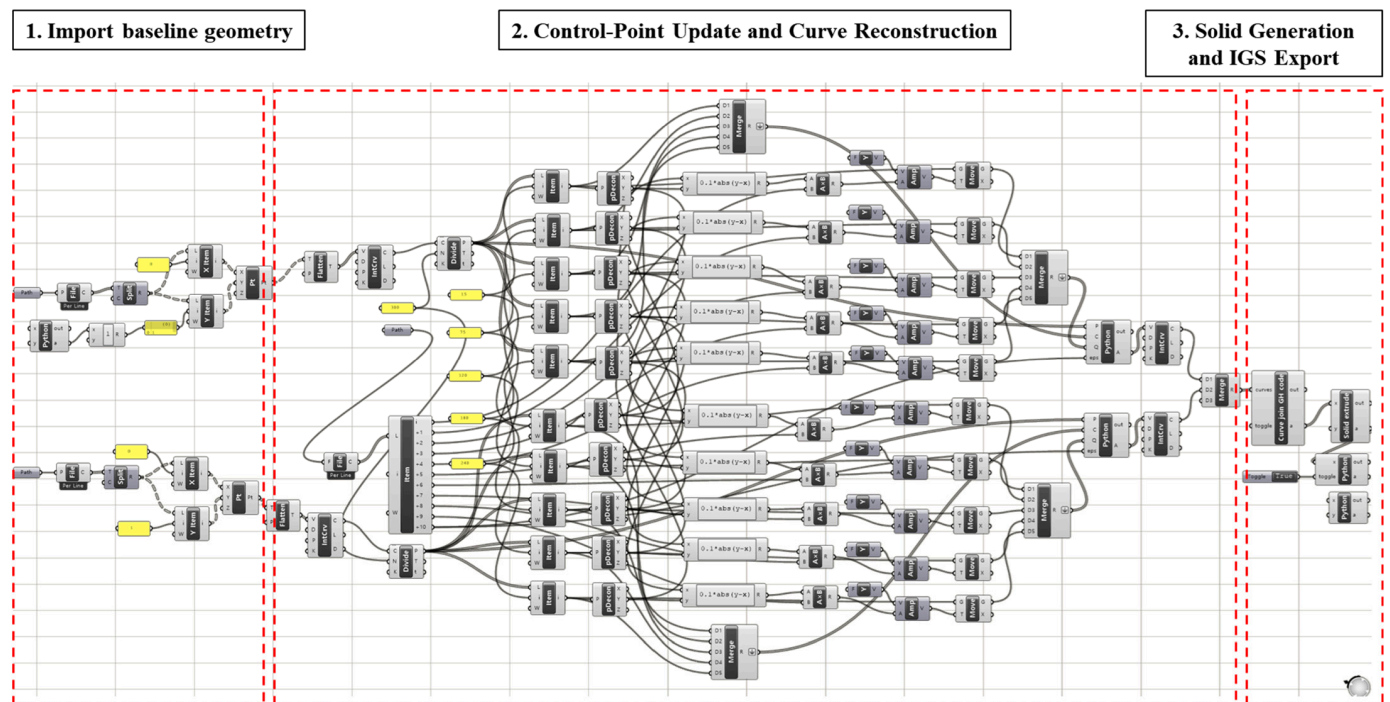


Figure 3. Geometry modification and solid generation process using grasshopper.

In Process 1, the baseline section geometry is imported and discretized into an ordered point list $P = \{p_k\}_{k=1}^N$, where $p_k = (x_k, y_k, z_k)$ denotes the k -th contour point and the discretization uses $N = 300$ points, and the ordering is preserved so that edge treatments are consistently applied at the same contour locations across iterations. In Process 2, the design update from HEEDS is introduced through a small set of control points $C = \{c_i\}_{i=1}^n$ and their updated target points $Q = \{q_i\}_{i=1}^n$ with $n = 5$, where $c_i = (c_{i,x}, c_{i,y}, c_{i,z})$ and $q_i = (q_{i,x}, q_{i,y}, q_{i,z})$, and only the vertical displacement is used as the interpolation data.

$$d_i^H = q_{i,y} - c_{i,y}, \quad i = 1, \dots, n. \quad (1)$$

To prevent excessive deformation, the Grasshopper definition applies a thickness based bound using paired upper and lower surface coordinates at the same chordwise location. Let y^U and y^L denote the upper and lower surface y coordinates, respectively. The local thickness is

$$t = |y^U - y^L| \quad (2)$$

The allowable displacement magnitude is limited to 10 percent of this thickness, following the expression implemented in the definition. This bound was applied as the main explicit geometric constraint in the present optimization. No separate explicit constraints were imposed on the leading-edge radius, trailing-edge angle, or maximum camber. These quantities were indirectly controlled by the fixed chordwise locations of the control points, the limited vertical displacement range, and the smoothing and edge-protection treatments described below. The HEEDS proposed displacement is then clamped within this bound before it is passed to the interpolation stage. The bounded control point displacements are propagated to the full contour using a thin plate spline radial basis interpolation with an affine term, implemented in the Python component. The thin plate spline (TPS) kernel is defined as

$$\phi(r) = (r^2 + \epsilon^2) \ln(r^2 + \epsilon^2) \quad (3)$$

where r is the Euclidean distance between two points and $\varepsilon > 0$ is a kernel smoothing parameter introduced for numerical robustness as $r \rightarrow 0$. In the implementation, $\varepsilon = \max(\text{eps}, 5 \times 10^{-5})$ and $\text{eps} = 10^{-4}$ is used when not specified. The displacement field $s(x, y)$, representing the propagated vertical displacement at a query location (x, y) , is expressed as:

$$s(x, y) = \sum_{i=1}^n \omega_i \phi(r_i(x, y)) + a_0 + a_1 x + a_2 y \quad (4)$$

where w_i are the RBF (Radial basis function) weights, (a_0, a_1, a_2) are affine coefficients, and the distance from (x, y) to control point $\mathbf{c}_i$ in the (x, y) plane is

$$r_i = \|(x, y) - (x_i, y_i)\| \quad (5)$$

The coefficients $\mathbf{w} = [w_1, \dots, w_n]^T$ and $\mathbf{a} = [a_0, a_1, a_2]^T$ are obtained by solving the regularized TPS linear system

$$\begin{bmatrix} K + \lambda I & P \\ P^T & 0 \end{bmatrix} \begin{bmatrix} \mathbf{w} \\ \mathbf{a} \end{bmatrix} = \begin{bmatrix} \mathbf{d} \\ \mathbf{0} \end{bmatrix} \quad (6)$$

where $\lambda = 10^{-6}$ is the diagonal regularization is used in the Python code, $\mathbf{d} = [d_1, \dots, d_n]^T$ is the bounded displacement vector, and the matrix blocks are defined by $K_{ij} = \phi(\|\mathbf{c}_i - \mathbf{c}_j\|)$ and $P_i = [1, c_{i,x}, c_{i,y}]$. The resulting TPS field is evaluated at each baseline contour point $\mathbf{p}_k = (x_k, y_k, z_k)$ and only the y-coordinate is updated.

For the leading edge, the first 8 points are smoothed using a quintic smoother step ramp to avoid abrupt deformation. The ramp function is below:

$$S(t) = 6t^5 - 15t^4 + 10t^3, \quad t \in [0, 1] \quad (7)$$

It is applied as a weight from 0 to 1 across the first 8 points, scaling the TPS displacement gradually from zero to the full value. Additionally, the first point is anchored to the baseline geometry. For the trailing edge, robustness is enforced by introducing a fixed zone and a taper zone. The last 12 points are fully fixed to the baseline geometry. Upstream of the fixed zone, a taper region of 60 points is used to smoothly reduce the deformation to zero based on a chordwise x-coordinate mapping, with an index-based fallback when the corresponding x-span becomes numerically small. This tapering ensures a continuous transition between the deformed mid-chord region and the fixed trailing edge, preventing the trailing edge from intersecting and maintaining a stable cross-section closure. In Process 3, the final point list is reconstructed as a solid geometry within Grasshopper and exported as an IGS file for STAR-CCM+ for subsequent mesh generation and CFD evaluation.

2.3. Numerical Setup

All CFD simulations were carried out using the commercial software STAR-CCM+ with a Reynolds averaged Navier Stokes formulation. The flow is modeled as incompressible, and the governing equations for the mean field are the continuity equation and the momentum equation given by

$$\nabla \cdot \mathbf{U} = 0 \quad (8)$$

$$\rho \left(\frac{\partial \mathbf{U}}{\partial t} + \mathbf{U} \cdot \nabla \mathbf{U} \right) = -\nabla P + \mu \nabla^2 \mathbf{U} - \rho \nabla \cdot \overline{\mathbf{u}'\mathbf{u}'} + \rho \mathbf{g} \quad (9)$$

where $\mathbf{U}$ and P denote the Reynolds averaged velocity and pressure, ρ is the fluid density, and μ is the dynamic viscosity. The Reynolds stress term $-\rho \overline{\mathbf{u}'\mathbf{u}'}$ derives from the

decomposition $u = U + u'$, and it is closed using the realizable $k - \epsilon$ turbulence model in this study.

The computational domain and boundary condition layout are shown in Figure 4a, where the far-field extents are defined relative to the section chord length to reduce blockage and boundary interference. The simulations are conducted in a quasi-two-dimensional configuration by extruding the 2D section in the spanwise direction and using a very small domain thickness ($0.01L_{Chord}$) in the z direction. Symmetry boundaries are applied on the spanwise planes so that variations across the thin z extent are suppressed and the solution represents a two-dimensional flow field while still being solved with the three-dimensional solver in STAR-CCM+. A velocity inlet is prescribed at the upstream boundary, and a pressure outlet is applied downstream, while the airfoil surface is treated as a no-slip wall. The remaining outer boundaries are set as symmetry planes as illustrated in the figure. In preliminary tests, a short upstream distance caused an overprediction of drag, which indicated inlet proximity effects. For this reason, the inlet boundary is placed $30L_{chord}$ upstream of the airfoil and the outlet boundary is placed $15L_{chord}$ downstream. The transverse clearance is set to $10L_{chord}$ above and below the section.

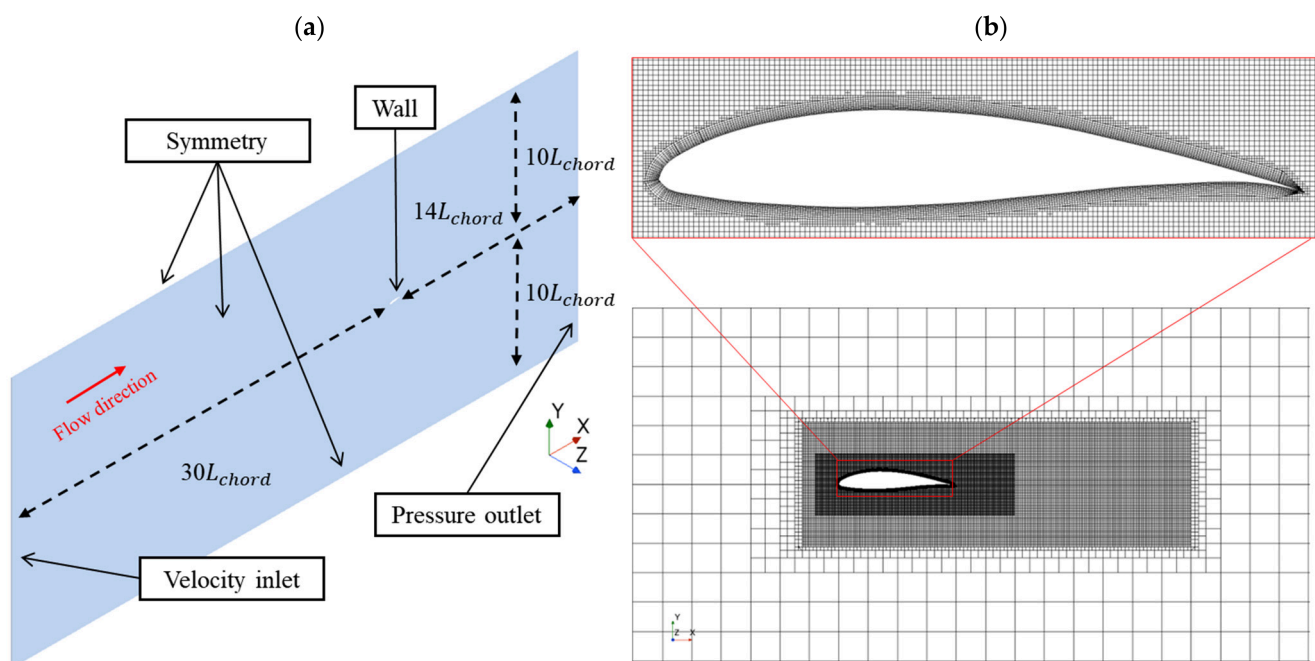


Figure 4. (a) Boundary condition and (b) mesh distribution for CFD simulation.

Mesh generation follows the setup summarized in Table 1. A trimmed cell mesh is used with automatic surface repair and surface remeshing, and prism layers are added near the airfoil to capture the near-wall flow. In addition to the near-wall refinement, the mesh is locally refined to resolve the wake development downstream of the trailing edge. As shown in Figure 4b, nested refinement regions are applied with very fine cells in the near-wake zone and fine cells in the surrounding downstream region, providing a gradual transition to the background mesh while maintaining sufficient resolution in the wake. The prism layer configuration uses 10 layers with a near-wall prism thickness of 0.001 m, a gap fill percentage of 25%, a minimum thickness percentage of 0.01%, and a boundary march angle of 85 degrees. The resulting mesh contains 88,963 cells. The flow is modeled as turbulent using the realizable $k - \epsilon$ model. This prism layer configuration was selected through preliminary near-wall mesh checks, where the predicted lift, drag, and surface pressure distribution were found to be sensitive to the prism-layer resolution. Several near-wall mesh settings were examined before selecting the present configuration, which

provided stable hydrodynamic coefficients and reproduced the main pressure-distribution trends observed in the reference data. The realizable k- ϵ turbulence model was adopted for its robust convergence behavior and numerical stability in repeated CFD calculations under consistent numerical settings. The solver uses an implicit unsteady formulation with a time step of 0.05 s and first-order temporal discretization. The maximum number of inner iterations is set to five per time step.

Table 1. Numerical set-up for airfoil simulation using STAR-CCM+.

Mesh generation	Mesher	Trimmed cell mesher
		Prism layer mesher
		- Number of prism layers: 10
		- Prism layer near wall thickness: 0.001 m
		- Gap fill percentage: 25%
		- Minimum thickness percentage: 0.01%
		- Boundary march angle: 85°
		Surface remesher
		Automatic surface repair
	Total number of cells	88,963
Physics setup	Space	Three dimensional
	Time	Implicit unsteady
	Viscous regime	Turbulent
	Turbulence model	Realizable K-ε model
Boundary conditions	Fluid region Boundary conditions	Inlet: Velocity inlet Outlet: Pressure outlet Airfoil surface: wall Top, bottom, two sides: symmetry
Solvers	Implicit unsteady	Time-step: 0.05 s Temporal discretization: 1st-order
Stopping criteria	Maximum inner iterations: 5	

2.4. Convergence Test and Validation

To assess the numerical reliability of the CFD results, convergence tests were performed for both grid resolution and time step size. Grid and time-step convergence were evaluated using the Grid Convergence Index (GCI) method. For a representative solution variable ϕ , the refinement ratio r , apparent order of accuracy p , extrapolated solution ϕ_{ext} , approximate relative error e_a , and GCI were calculated as follows:

$$p = \frac{\ln \left| \frac{\phi_3 - \phi_2}{\phi_2 - \phi_1} \right|}{\ln(r)}, \quad r = \frac{h_2}{h_1} \quad (10)$$

$$\phi_{\text{ext}} = \frac{r^p \phi_1 - \phi_2}{r^p - 1} \quad (11)$$

$$e_a = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right| \quad (12)$$

$$GCI = \frac{F_s e_a}{r^p - 1} \times 100 \quad (13)$$

Here, ϕ_1 , ϕ_2 , and ϕ_3 denote the solutions obtained from the fine, medium, and coarse grids, respectively. The safety factor F_s was set to 1.25 for three-level refinement. The same

procedure was applied to the time-step convergence test by replacing the grid size h with the time-step size Δt .

A grid convergence study was carried out using three grid levels generated by scaling the base size with a constant ratio of 1.4 between cases. This procedure resulted in meshes of 42,423, 88,963, and 154,347 cells while keeping the time step at 0.05 s. As reported in Table 2, the predicted lift increased from 287.5 N to 290.1 N with grid refinement, while the drag decreased from 4.01 N to 3.98 N. The corresponding GCI values for the medium grid case G2T2 were 1.16% for lift and 0.79% for drag, indicating limited sensitivity to further refinement. On the other hand, a time step convergence test was then performed on the medium grid of 88,963 cells using time steps of 0.10 s, 0.05 s, and 0.01 s. The lift changed from 287.7 N to 289.5 N and the drag changed from 4.05 N to 3.97 N as the time step was reduced. The GCI values for the reference case G2T2 were 0.03% for lift and 0.32% for drag. A GCI level below 2% was adopted as the acceptance criterion for the representative lift and drag quantities in the present comparative optimization. Based on these results, the medium grid and the 0.05 s time step were selected for the optimization runs as they provided stable predictions with acceptable numerical uncertainty.

Table 2. Comparison of GCIs of the NACA 63–815.

	Case No.	Number of Grids	Δt	Lift(N)	GCI (%)	Drag(N)	GCI (%)
Grid	G1T2	42,423	0.05	287.5	1.16	4.01	0.79
	G2T2	88,963	0.05	289.2		3.99	
	G3T2	154,347	0.05	290.1		3.98	
Time step	G2T1	88,963	0.10	287.7	0.03	4.05	0.32
	G2T2	88,963	0.05	289.2		3.99	
	G2T3	88,963	0.01	289.5		3.97	

Validation and benchmark comparisons were performed using two published references shown in Figure 5. The first reference is the experimental and XFOIL data of Molland et al. (2004) [23] at $Re = 8 \times 10^5$, which provide measured lift and drag polars for the NACA 63–815 section. Because this Reynolds number is lower than that of the present simulations, the data are used as a lower Reynolds number experimental reference rather than as a complete validation dataset for $Re = 1 \times 10^7$. The second reference is the XFOIL data of Baltazar et al. (2011) [24] at $Re = 1 \times 10^7$, which provides the same Reynolds number of numerical benchmarks for the present simulations.

Compared with the experiments of Molland et al. (2004) [23], the present CFD results show some differences, which are expected because Reynolds-number effects influence boundary-layer development, skin friction, separation onset, and drag prediction. When the comparison is made against the Baltazar et al. results at $Re = 1 \times 10^7$, the lift coefficient shows close agreement over the angle-of-attack range investigated. The drag coefficient follows a similar trend at low to moderate angles of attack but deviates as the angle increases. This discrepancy is attributed to fundamental differences between XFOIL and RANS-based CFD. XFOIL combines a potential-flow panel method with an integral boundary-layer model, while the present simulations solve the RANS equations with a turbulence model to represent boundary-layer growth, separation, and wake development. Therefore, larger differences in drag can occur at higher incidence, where the result becomes more sensitive to transition, separation, and wake behavior.

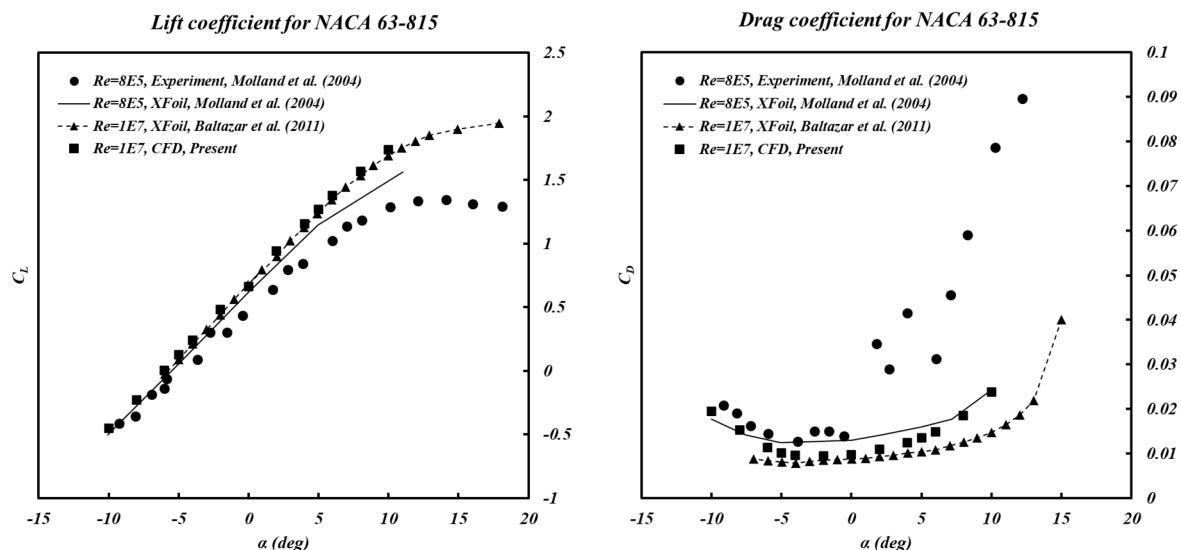


Figure 5. Comparison of lift coefficient and drag coefficient for CFD validation.

Accordingly, the present CFD setup was considered appropriate for the subsequent comparative optimization. The objective of this study is to evaluate relative performance differences among candidate sections under identical numerical conditions, rather than to certify absolute performance at a single operating point. Within this purpose, the agreement in lift trends and the consistent drag behavior over the main angle-of-attack range provide sufficient support for using the CFD results in the optimization process.

3. Result Analysis

To obtain an optimized blade section, a total of 1000 design evaluations were carried out in the HEEDS optimization loop. The section geometry was parameterized using 10 control points, five on the suction side and five on the pressure side. In HEEDS, the design variables were defined as the normalized vertical displacements of these points within the range from -1 to 1 , which corresponds to allowing each point to move within plus or minus 10 percent of the local section thickness at its chordwise location. This bound prevents unrealistic deformations while retaining sufficient flexibility to adjust the camber and thickness distribution. For each candidate proposed by SHERPA, the updated geometry was regenerated in Rhino Grasshopper and exported as an IGS file. The CFD simulations were then executed automatically through a JavaScript based workflow.

3.1. Trend Analysis

The prescribed control point displacements alter the local camber and thickness, which in turn affects the pressure distribution and near wall flow development at the Reynolds numbers relevant to tidal stream turbines. Since the effective inflow angle at a blade section varies with current fluctuations, turbulence and shear, and induction effects, and may further change under yaw misalignment or platform motions in floating concepts, the design was evaluated over a small set of representative angles of attack. The objective angles of 0 deg, 3 deg, 6 deg, and 9 deg were selected to cover the main positive effective angle-of-attack range relevant to tidal turbine blade-section operation. This range represents attached to moderately loaded conditions before strong stall behavior and was used to avoid single-point optimization. It does not represent a full motion envelope of a specific floating platform, but accounts for practical variations in effective angle of attack caused by inflow variation, induction effects, yaw misalignment, and possible platform motion. This range corresponds to attached flow to moderately loaded conditions, where

maintaining lift generation, sectional efficiency, and constrained drag is important for normal turbine operation.

For each evaluation, four angles of attack, 0 deg, 3 deg, 6 deg, and 9 deg, were simulated, and the corresponding lift and drag coefficients, C_L and C_D , were returned to HEEDS as responses. Based on these responses, two performance metrics were computed in HEEDS using formula expressions. The first metric was $C_{D,max}$, defined as the maximum drag coefficient among the four angles, and it was minimized to control the worst-case loss within the considered operating range. This choice is particularly relevant for tidal stream turbines because the effective angle of attack at a blade section varies in time due to current fluctuations, turbulence and shear, it may further deviate under yaw misalignment or platform motions in floating concepts. Under such conditions, a section that performs well at a single angle can still produce a pronounced drag rise at another angle, which typically coincides with stronger separation and leads to a disproportionate drop in hydrodynamic efficiency. Therefore, the maximum drag penalty helps to filter out geometries that are sensitive to situations outside the design conditions, reducing the possibility of unfavorable drag variations that can increase losses and lead to higher thrust and bending loads. The second metric was the mean lift to drag ratio, computed by averaging C_L/C_D across all four angles, and it was maximized to promote consistently high sectional efficiency across the same range. In practical turbine design, this encourages a section that maintains favorable loading with relatively low drag over the expected variability in inflow angle, supporting stable energy capture while avoiding designs that rely on narrow, single point tuning.

To get optimized design, the performance index combines C_L/C_D and $C_{D,max}$ after min max normalization over the feasible set. The final design was selected as the best compromise solution that minimizes the normalized distance to the ideal point, which corresponds to maximum C_L/C_D and minimum $C_{D,max}$. The SHERPA optimization was performed with 1000 design evaluations, considering the practical computational cost of repeated CFD simulations. Since each candidate required CFD calculations at four angles of attack, this evaluation number was selected to provide a reasonable balance between design-space exploration and computational efficiency. Figure 6 summarizes both the convergence behavior and the objective trade-off observed in the HEEDS study. Since the study is formulated as a two-objective optimization, a scalar performance index was introduced only for convergence visualization and design ranking. In Figure 6a, the objective values of all feasible designs are plotted against the evaluation index, and the dotted step line tracks the best-so-far performance. The best performance rises rapidly during the initial stage, reaching approximately 0.24 within the 50 to 100 evaluations. Subsequent designs continue to span a wide range of performance, but the best-so-far line remains essentially unchanged over a long interval, indicating a clear plateau. A final improvement occurs at Design 746, with a performance of about 0.25, and no further improvement is obtained during the remaining evaluations. Considering the computational efficiency required for repeated CFD-based design evaluation, Design 746 was selected as a satisfactory best feasible solution for the present study. Figure 6b explains this outcome in terms of the response metrics. The scatter plot of $C_{D,max}$ versus C_L/C_D shows a distinct trade-off, where lower $C_{D,max}$ is generally associated with higher hydrodynamic efficiency. Design 746 lies close to the upper-left boundary of the distribution, demonstrating one of the lowest $C_{D,max}$ levels while maintaining a high C_L/C_D . Although designs with slightly higher C_L/C_D exist, they typically incur higher $C_{D,max}$, which clarifies why Design 746 emerges as a balanced, high-performing candidate within the feasible set under the imposed bounds and constraints.

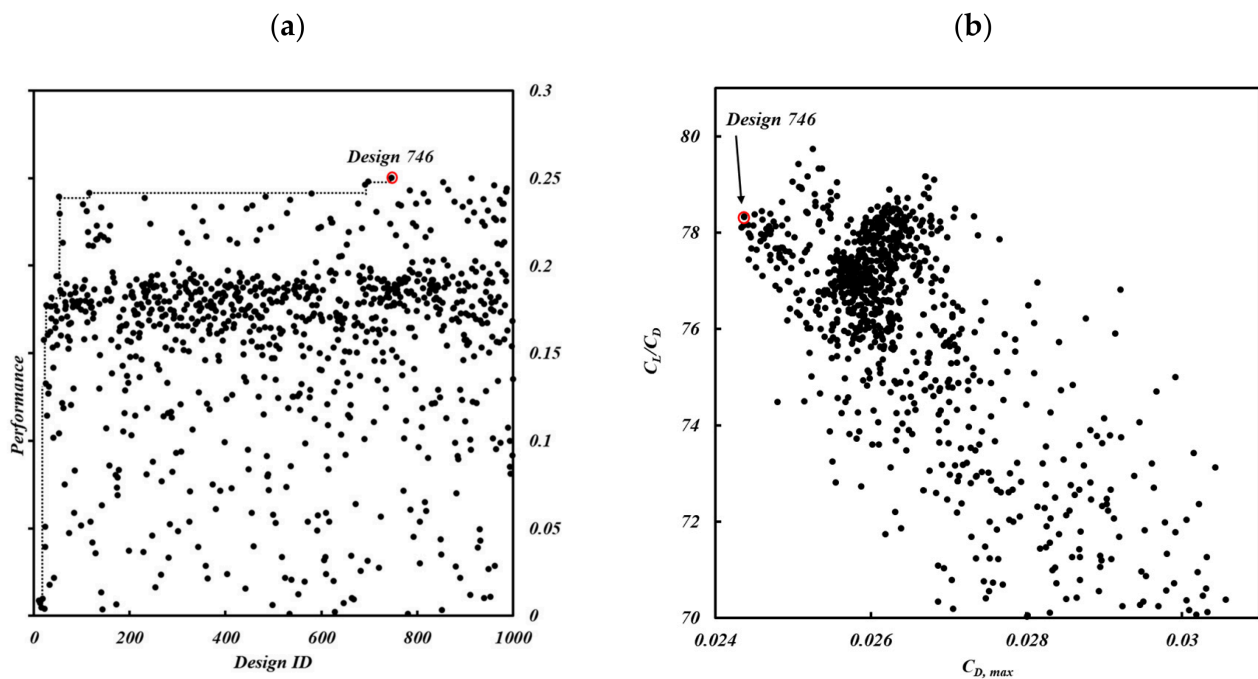


Figure 6. Optimization results from the HEEDS (a) Performance index, (b) $C_{D,max}$ vs. C_L/C_D .

Figure 7 compares the hydrodynamic responses, which compare dimensionless performance coefficients, including lift, drag, and efficiency, across the evaluated designs at $\alpha = 0^\circ, 3^\circ, 6^\circ$, and 9° . In each panel, the full set of feasible designs forms a broad envelope, illustrating the range of achievable performance under the prescribed design variable bounds. The baseline and the selected best design are overlaid to show how the final choice is positioned within this distribution. In Figure 7a, the best design follows the expected trend of increasing C_L with angle of attack and remains within the upper range of the feasible envelope across all angles, indicating that the drag constraint imposed in the optimization is not accompanied by an anomalous loss of lift. The baseline curve lies closer to the central region of the distribution, whereas the best design shifts toward the high C_L , particularly at the larger angles, suggesting that the optimized geometry sustains lift generation over the operational range considered.

Figure 7b highlights the key factor behind the selected solution. The best design is located near the lower boundary of the C_D envelope at all four angles, indicating a consistent suppression of drag and a constrained $C_{D,max}$ rather than an improvement limited to a single operating point. The baseline curve is generally located at higher C_D , and the separation between the baseline and the best design becomes most evident at the higher angles, where drag penalties are typically more pronounced. These combined effects are reflected in Figure 7c. The best design occupies the upper region of the C_L/C_D envelope across the full angle set, demonstrating that the improvement is not concentrated at one angle but maintained over the range used to define the objective metrics. While some candidates exceed the best design at a specific angle, those designs typically correspond to higher C_D in Figure 7b, consistent with the trade-off observed in the response space. Overall, the three panels indicate that the selected design achieves a balanced improvement characterized by constrained drag and high efficiency across the considered angles of attack.

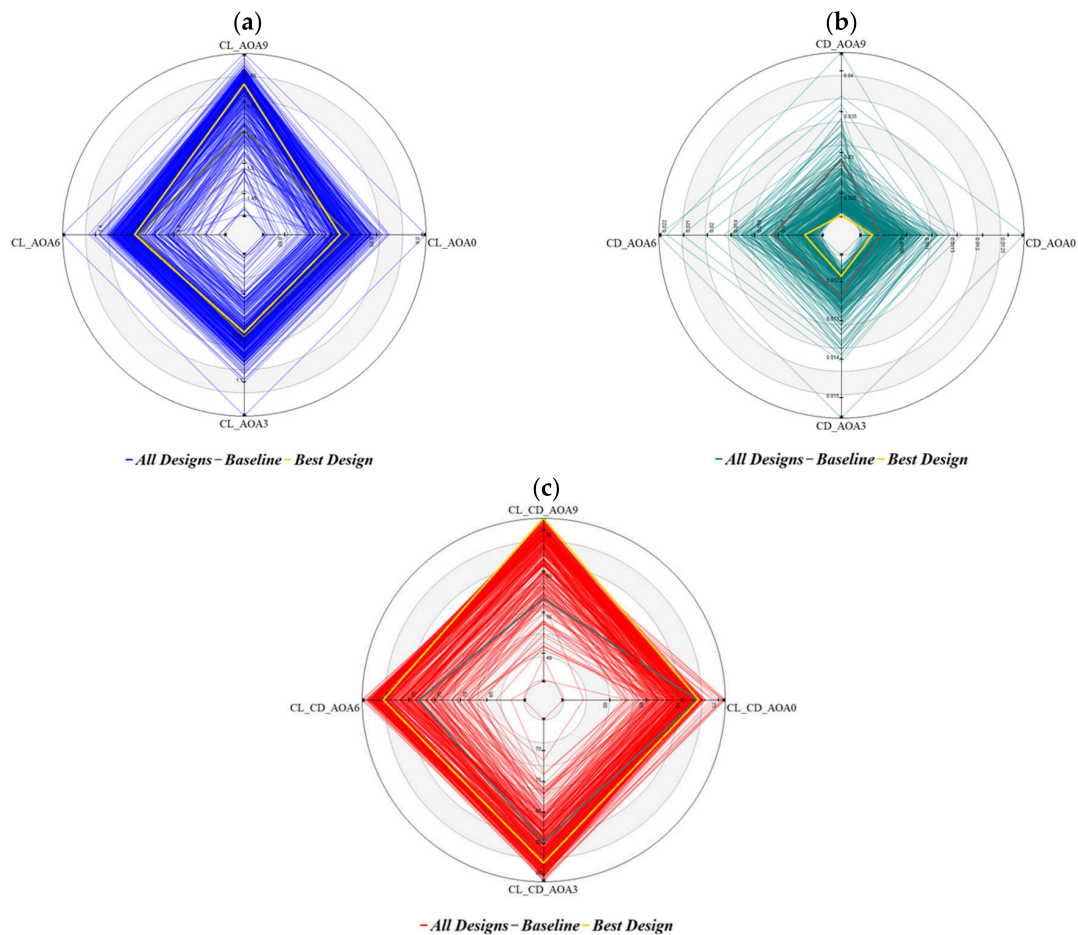


Figure 7. Radar plots for (a) lift coefficient C_L , (b) drag coefficient C_D , and (c) efficiency C_L/C_D .

3.2. Hydrodynamic Performance Analysis

In this chapter, the hydrodynamic performance of the original and optimized sections was quantitatively compared under identical conditions. Based on CFD analysis, the lift coefficient, drag coefficient, and lift-to-drag ratio were compared according to the angle of attack. In addition, the minimum pressure, cavitation inception characteristics, and the angle-dependent surface pressure distribution were examined to identify the physical causes of the performance changes.

Figure 8 compares the original NACA 63–815 section with optimized geometry. Relative to the baseline, the optimized shape is formed by a coordinated redistribution of the control point displacements, rather than a uniform thickening or thinning. The upper surface control points in the fore chord are shifted outward, increasing the local suction-side height and promoting a more forward-loaded pressure distribution. In contrast, the upper surface in the aft region is slightly pulled inward, which relaxes the rear suction-side curvature and avoids excessive aft loading. On the pressure side, the front control points move upward, reducing the leading-edge concavity of the lower surface, while the mid-to-aft control points move downward to recover thickness and camber where it contributes to lift without strongly penalizing the trailing-edge behavior. Because these adjustments are applied in a balanced manner across the two surfaces, the resulting section exhibits a forward redistribution of thickness and camber with smoothly blended transitions, while maintaining essentially unchanged trailing-edge closure. Overall, the optimized geometry reflects a targeted reshaping strategy that concentrates the main modification near the leading-edge to mid-chord region and keeps the global characteristics of the baseline profile intact.

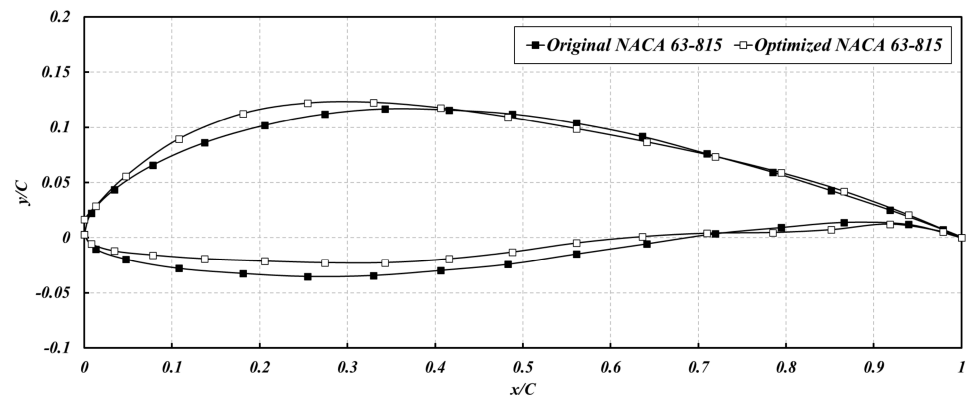


Figure 8. Comparison of the original and optimized section geometries for NACA 63–815.

Figure 9 compares the sectional performance of the original and optimized NACA 63–815 at $Re = 1 \times 10^7$. After the optimization, both sections were evaluated over the full angle range shown in the figure to examine the overall performance trend. In Figure 9a, the lift coefficient increases with angle of attack for both sections, while the optimized section provides higher lift across the full range. Relative to the original, C_L increases by 7.2 to 14.5 percent in the low angle range from -4° to 0° , and by about 2.1 to 3.4 percent in the operating range from 2° to 10° . The drag coefficient remains at a similar level, showing only small changes of about -3.7 to $+6.2$ percent depending on angle, and no noticeable penalty at the peak region. These trends are consistent with Figure 9b, where the optimized section maintains a higher C_L/C_D . Compared to the original, C_L/C_D improves by about 2.1 to 14.5 percent, with the largest gains at -4° and 0° , and sustained improvements near the region where C_L/C_D reaches its maximum. Overall, the optimized section improves the average hydrodynamic efficiency under the present conditions while keeping drag growth effectively constrained.

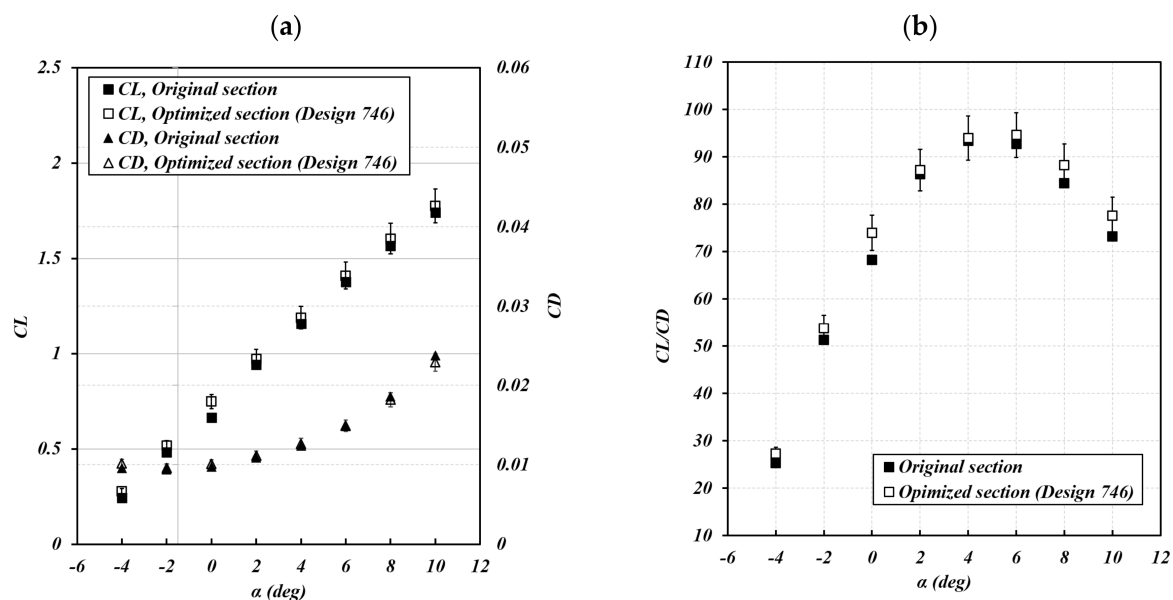


Figure 9. Sectional performance polars of the original and optimized NACA 63–815. (a) lift and drag coefficient, (b) lift-to-drag ratio.

Figure 10 shows the relationship between the cavitation number σ and the sectional lift coefficient C_L , which represents cavitation performance from the perspective of inception. The cavitation number is defined as $\sigma = (p_\infty - p_v) / (0.5\rho U^2)$, and cavitation inception is identified when the minimum pressure on the airfoil surface first reaches the vapor pressure

under each operating condition. For each condition, σ is obtained from the surface pressure field computed by CFD and plotted together with C_L . Compared with the original section, the optimized shape shifts the inception envelope slightly toward lower σ over the mid to high lift range, which indicates delayed cavitation inception under the same C_L criterion. This means that, for a comparable lift level, the optimized section reaches the minimum-pressure inception condition at a lower cavitation number than the original section. At the same time, the optimized section sustains a higher C_L level for a given σ in the upper part of the envelope, implying a wider cavitation free margin despite the increased loading. Considering the operating range of this study, these results suggest that optimized geometry provides a more favorable inception characteristic through moderated minimum pressure behavior. Ge et al. (2022) [25] showed that hydrodynamic cavitation development is affected by cavitation number, Reynolds number, and thermodynamic effects. Similarly, the present result indicates that the moderated minimum-pressure behavior and smoother pressure recovery of the optimized section can contribute to delayed cavitation inception.

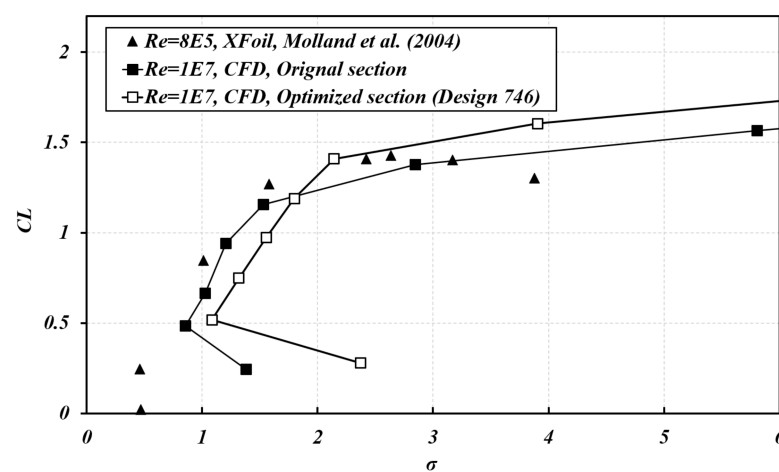


Figure 10. Cavitation inception envelope of the original and optimized NACA 63–815 [24].

However, the present cavitation assessment is limited to an indirect comparison of inception tendency based on the minimum surface pressure criterion. Full multiphase cavitation simulation and experimental validation of cavitation inception were not performed in this study. Therefore, the cavitation bucket should be interpreted as a relative comparison between the original and optimized sections under identical numerical conditions, rather than as a fully validated prediction of cavitation performance. In addition, the trend of the original section shows reasonable agreement with the reference data shown in the figure, supporting the qualitative consistency of the present inception assessment.

Figure 11 compares the surface pressure coefficient distributions of the original and optimized NACA 63–815 at three angles of attack, revealing the pressure field changes that underlie the lift, drag, and cavitation inception trends discussed earlier. Each panel presents the $-C_p$ distribution for $\alpha = 0^\circ$, $\alpha = -5.2^\circ$, $\alpha = 6.8^\circ$ versus x/c . The CFD results for the original and optimized sections are shown together with the experimental reference data reported by Molland et al. (2004) [23]. Overall, the original configuration reproduces the experimental trends reasonably well, particularly in the leading-edge suction peak and the subsequent pressure recovery, which supports the reliability of the present CFD setup for comparing the two geometries. At $\alpha = 0^\circ$, both sections show similar recovery behavior on the suction and pressure surfaces, and the differences between the two CFD curves remain modest across most of the chord.

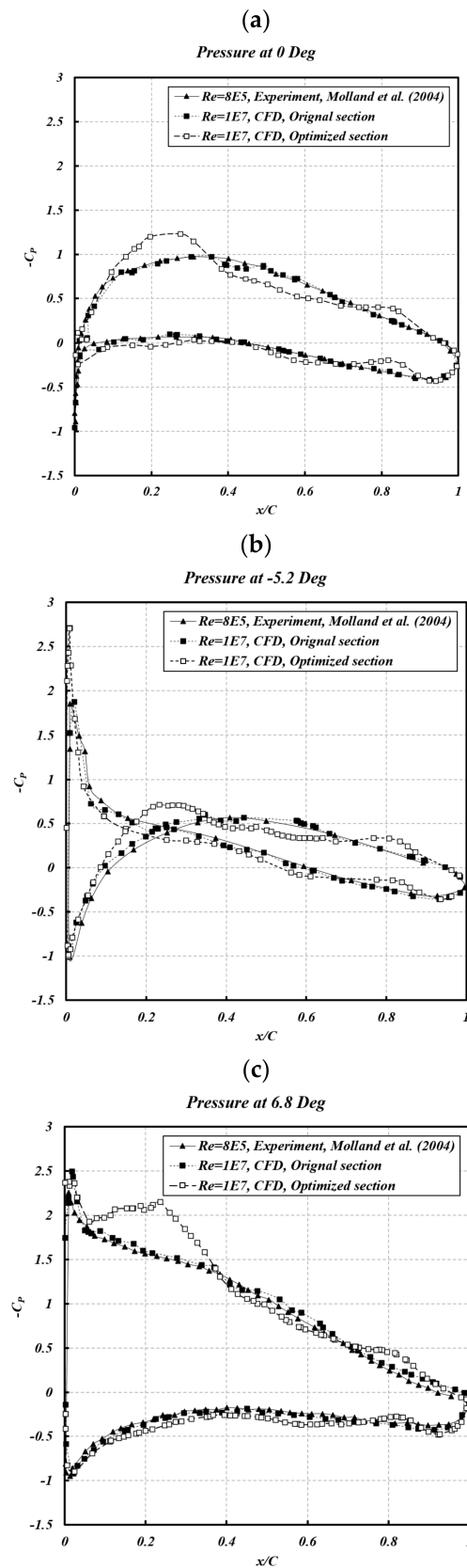


Figure 11. Comparison of pressure coefficient distributions for the original and optimized NACA 63–815 [24].

At $\alpha = 0^\circ$, the two sections show similar pressure side levels, while the optimized section exhibits a slightly different suction side profile in the fore chord. The suction side is

marginally higher in the front to mid region, with a small relaxation toward the aft chord, indicating that the optimization mainly redistributes the loading rather than increasing it uniformly. At $\alpha = -5.2^\circ$, the pressure distribution becomes more sensitive near the leading edge. The optimized section shows a noticeable change in the leading-edge region and a smoother recovery trend over the mid chord, suggesting a more concentrated but balanced loading pattern under negative incidence. This behavior is consistent with the tendency for performance differences to emerge more clearly at off design angles. At $\alpha = 6.8^\circ$, the suction side dominates the overall distribution, and the optimized section maintains slightly stronger suction over the fore chord, while the pressure side remains close to the original. The difference is most evident near the front part of the chord, where the optimized curve indicates a modified suction level and recovery pattern. This explains the higher lift observed for the optimized geometry at the same angle of attack. The leading-edge suction peak and the following pressure recovery region are directly related to the critical low-pressure region associated with cavitation inception. Therefore, the smoother recovery pattern of the optimized section supports the delayed inception tendency observed in Figure 10.

3.3. 3D Performance Comparison & Discussion

In this section, the three-dimensional rotor performance was examined to determine whether the improvement obtained from the two-dimensional section optimization could be translated into an actual rotor-level benefit. This comparison was performed using full 3D RANS CFD simulations in STAR-CCM+, rather than a BEM-based calculation using inserted 2D polar curves. For this purpose, two three-dimensional tidal turbine rotors were constructed under identical conditions by applying the optimized 2D foil developed in the present study and the baseline NACA 63–815, respectively. The rotor geometry, including chord, twist, and radial thickness distributions, was adopted from Zhu et al. (2020) [15], where the rotor performance was validated against experimental fluid dynamics data. Therefore, the present 3D analysis should be interpreted as a preliminary comparison using a prescribed rotor geometry rather than as a complete rotor-level optimization. The radial distributions of chord, twist, and thickness used for the blade construction are summarized in Table 3. Because of this limited purpose, a separate 3D mesh-convergence study and y^+ sensitivity assessment were not included in the present manuscript. The 3D results are therefore interpreted only as rotor-level performance tendencies under the prescribed geometry.

The applied three-dimensional rotor geometry is shown in Figure 12. Approximately 1.5 million cells were used for the full 3D RANS CFD simulations, and the realizable $k - \epsilon$ turbulence model adopted in the 2D calculations was retained for consistency.

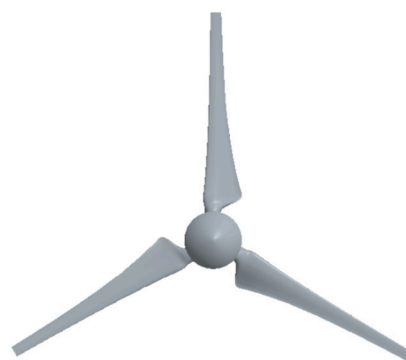


Figure 12. Geometry of 3D rotor.

Table 3. Blade geometry parameter distribution.

r/R	r (m)	c (m)	β (deg)	t/c (%)
0.20	0.540	0.4878	21.1	30.0
0.25	0.675	0.4257	17.4	29.1
0.30	0.810	0.3744	14.5	28.1
0.35	0.945	0.3321	12.1	27.2
0.40	1.080	0.2979	10.2	26.3
0.45	1.215	0.2691	8.6	25.3
0.50	1.350	0.2457	7.3	24.4
0.55	1.485	0.2250	6.1	23.4
0.60	1.620	0.2079	5.2	22.5
0.65	1.755	0.1926	4.4	21.6
0.70	1.890	0.1800	3.7	20.6
0.75	2.025	0.1683	3.1	19.7
0.80	2.160	0.1584	2.5	18.8
0.85	2.295	0.1494	2.0	17.8
0.90	2.430	0.1413	1.6	16.9
0.95	2.565	0.1341	1.2	15.9
1.00	2.700	0.1278	0.9	15.0

As shown in Figure 13, the computational domain and mesh refinement regions were revised to provide clearer information on the domain size and local mesh arrangement. The inlet, outlet, symmetry boundaries, rotor wall, and sliding mesh interface are indicated in the figure. The figure also shows the characteristic dimensions of the domain and the fine and medium refinement regions around the rotor and downstream wake. The cell size gradually increased toward the far-field domain. This meshing strategy was intended to resolve both the local flow variation near the blade surfaces and the global wake structure with sufficient accuracy. In addition, the time-step size was sequentially reduced according to the rotor angular displacement, from 20 deg to 10 deg, 5 deg, and 2 deg per step, and the simulation proceeded to the next time-step level once the hydrodynamic forces were sufficiently converged at the current level.

Figure 14 compares the power coefficients of the baseline NACA 63–815 rotor and the rotor using the optimized 2D foil over a range of TSR (Tip-speed ratio) values. The experimental data reported by Zhu et al. (2020) [15] are also included as a reference for the baseline rotor. The optimized rotor performed better mainly in the intermediate TSR range, which corresponds to the main operating region. The improvement was most noticeable from TSR 3 to 7, with increases of about 4.5%, 8.4%, 3.7%, 1.8%, and 2.4%, respectively. The largest improvement appeared at TSR 4, while the gain near the peak-performance region around TSR 6 was smaller but still positive. By contrast, the difference was negligible at TSR 2 and 8. At higher TSR, however, the trend was reversed. The optimized rotor showed a 7.3% lower power coefficient at TSR 9 and a 19.6% lower value at TSR 10. At TSR 11, the power coefficient dropped to nearly zero. This sharp decrease should not be interpreted as a beneficial design feature. Rather, it indicates that the section-level improvement obtained within the representative 2D angle-of-attack range does not necessarily carry out to all rotor operating conditions. At high TSR, local separation, tip effects, spanwise flow, and numerical sensitivity may become more influential, and further dedicated 3D analysis is required.

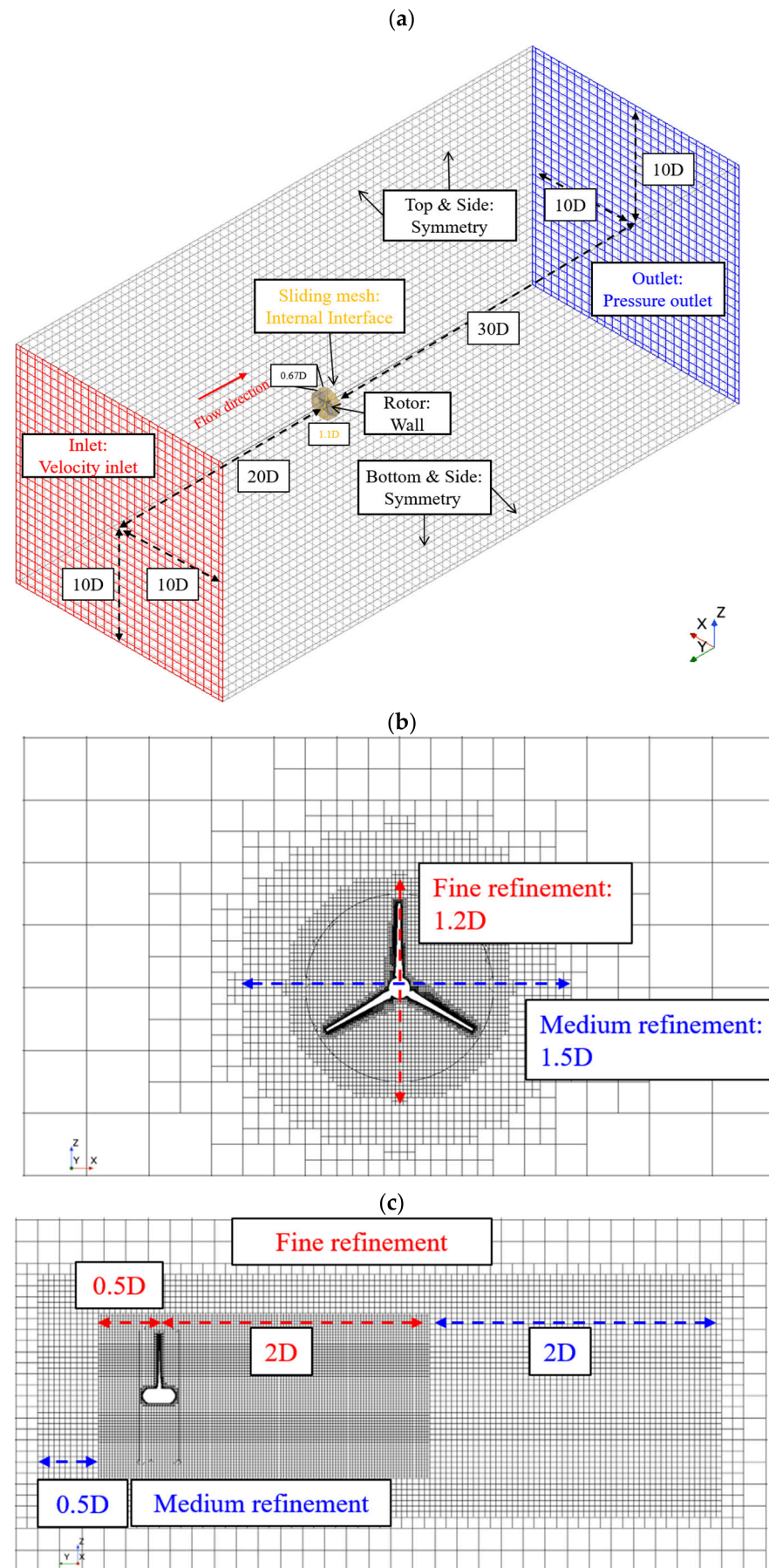


Figure 13. Grid distribution for 3D computation (a) Boundary conditions, (b) Front view, (c) Side view.

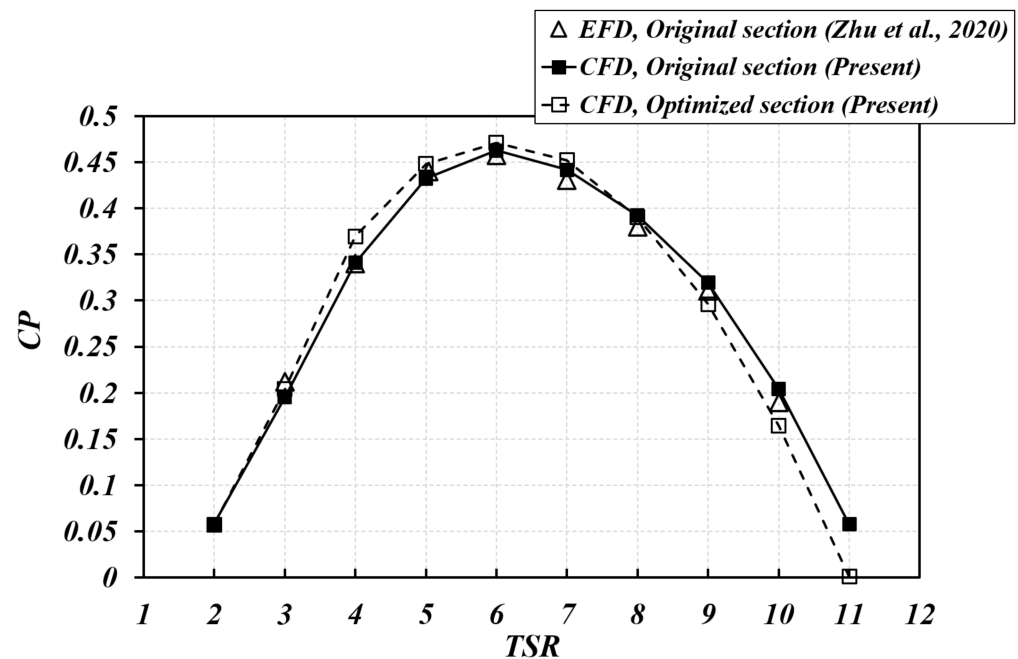


Figure 14. Comparison of power coefficients of 3D rotors [15].

The optimized section produced higher lift over the examined angle-of-attack range while keeping the drag level close to that of the baseline section. Lift increased by 7.2 to 14.5% at low angles of attack and by 2.1 to 3.4% over the main operating range, leading to an overall improvement in lift-to-drag ratio. The optimized section also showed delayed cavitation inception. These section-level improvements are consistent with the better performance observed in the mid-TSR region of the 3D rotor. The higher sectional lift and improved sectional efficiency appear to have contributed favorably to the spanwise loading, resulting in improved energy extraction over the main operating range. At high TSR, however, the performance of the optimized rotor deteriorated more rapidly. This suggests that the 2D optimization was effective within the representative angle-of-attack range considered in the section design stage, but its benefit did not fully carry over to all rotor operating conditions. In the actual rotor, three-dimensional effects such as tip loss, rotational augmentation, spanwise flow, and local separation become more important. For this reason, improvements at the section level do not necessarily lead to the same degree of gain at the rotor level. The selected design should therefore be understood as a section-level compromise that improves lift and efficiency under constrained drag, rather than a fully optimized 3D rotor with simultaneously optimized chord, twist, and radial loading distribution. Further improvement at the 3D rotor level will be addressed in future work through a fully coupled optimization of blade geometry and rotor operating characteristics.

In addition to the rotor-level limitations discussed above, the present study also has a limitation in terms of computational cost comparison. The computational cost of the automated CFD procedure depends on the number of candidate sections, mesh resolution, convergence settings, and available computing resources. Therefore, a complete cost comparison with other optimization procedures was not attempted, since such a comparison would require the same design problem, accuracy target, sampling strategy, and computational environment. The focus of the present study was placed on establishing a direct CFD-based design procedure in which candidate sections are evaluated under consistent numerical conditions.

4. Conclusions

This study presented an automated HEEDS-based workflow for CFD-driven design and evaluation of 2D tidal-stream turbine blade sections. The main contribution of this study is the development and demonstration of an automated CFD-driven design procedure for comparative tidal turbine blade-section assessment. The procedure integrates parametric geometry generation, CFD-based evaluation, and optimization in a reproducible loop. It was tried to derive an improved section efficiently in the early design stage by using a consistent and higher-fidelity evaluation procedure, while maintaining a clear physical interpretation of the resulting performance changes. The automation was constructed by linking HEEDS with Rhino Grasshopper for parametric geometry regeneration and with STAR-CCM+ for scripted CFD execution and numerical result extraction, so that candidate sections could be generated and evaluated in a closed loop under fixed numerical settings.

Using the NACA 63–815 section, a total of 1000 design evaluations were performed in the HEEDS optimization loop using a 10 control-point parameterization. Each evaluation included CFD simulations at four representative angles of attack, 0 deg, 3 deg, 6 deg, and 9 deg, and the responses were used to compute two trend-based metrics, the mean lift-to-drag ratio across the angle set and the maximum drag coefficient within the same set. The optimization history showed a rapid improvement in the early stage followed by a long plateau, and the final best solution was obtained at Design 746. The response-space distribution indicated a clear trade-off between the maximum drag penalty and sectional efficiency, and the selected design was positioned near the high-performance boundary as a balanced compromise.

Hydrodynamic performance was then compared between the original and optimized sections under identical conditions. The results showed that the optimized section produced higher lift across most of the investigated angle-of-attack range, while the drag level remained comparable to that of the original section without an increase in the peak drag. Accordingly, the improvement can be characterized as a lift gain achieved under a constrained maximum-drag behavior, which is consistent with the design objective formulation. The section-level benefit was confirmed through an overall increase in lift-to-drag ratio within the operating range used to define the mean-efficiency metric. Cavitation inception characteristics also improved in the sense that the optimized section tended to delay cavitation initiation under the same lift criterion, or to sustain a cavitation-free state at a higher lift under the same cavitation number criterion. This behavior indicates an improved inception margin associated with moderated minimum pressure characteristics under the present loading conditions. The pressure coefficient distributions supported this interpretation by showing that the geometry modification primarily altered suction-side loading in the front to mid-chord region and modified the pressure recovery behavior, which explains the observed trends in lift, efficiency, and inception margin. A preliminary three-dimensional rotor comparison further indicated that the optimized section could improve rotor power performance in the main operating TSR range under the prescribed rotor geometry. However, the benefit diminished at high TSR, showing that the present section-level optimization should be interpreted as a basis for rotor-level design exploration rather than as complete three-dimensional rotor optimization.

This study suggests that the proposed automated section-level workflow can provide a practical and repeatable basis for rotor development, particularly for floating tidal concepts where the effective inflow angle can vary and off-design behavior can influence losses and cavitation risk. When extended to three-dimensional blades, the same automation concept is expected to support more systematic exploration of blade geometries using consistent CFD settings and to improve traceability from geometry changes to rotor-level efficiency and cavitation margin. Further improvement at the three-dimensional rotor

level will be pursued in future work through full rotor-level optimization including blade geometry and operating-condition effects. Future work will apply the proposed automation procedure to three-dimensional blade geometries and rotor-level optimization and will consider additional operating conditions and constraints that reflect three-dimensional effects and practical design requirements. Experimental verification of the CFD-based results should also be included in future work, particularly for cavitation-related flow structures, and advanced multiphase flow measurement techniques such as synchrotron X-ray-based PIV and densitometry may provide useful validation data for this purpose.

Author Contributions: Conceptualization, S.L. and H.K.; Methodology, S.L. and H.K.; Investigation, S.K.; Resources, S.L. and S.K.; Data curation, S.L. and H.K.; Writing—original draft preparation, S.L.; Writing—review & editing, H.K. and S.K.; Visualization, S.L.; Supervision, H.K.; Funding acquisition, S.K. All authors have read and agreed to the published version of the manuscript.

Funding: This research was supported by Korea Institute of Marine Science & Technology Promotion (KIMST) Funded by the Ministry of Oceans and Fisheries (RS-2022-KS221571).

Data Availability Statement: The original contributions presented in the study are included in the article, further inquiries can be directed to the corresponding author.

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

References

1. Charlier, R.H. Sustainable Co-Generation from the Tides: A Review. *Renew. Sustain. Energy Rev.* **2003**, *7*, 187–213. [\[CrossRef\]](#)
2. Khojasteh, D.; Shamsipour, A.; Huang, L.; Tavakoli, S.; Haghani, M.; Flocard, F.; Farzadkhoo, M.; Iglesias, G.; Hemer, M.; Lewis, M.; et al. A Large-Scale Review of Wave and Tidal Energy Research over the Last 20 Years. *Ocean Eng.* **2023**, *282*, 114995. [\[CrossRef\]](#)
3. Lee, S.; Kim, H. Bibliometric Analysis of Tidal Energy: Trends in Tidal Turbine Design, Resource Assessment, Turbulence, and Feasibility Analysis. *Int. J. Nav. Archit. Ocean Eng.* **2025**, *17*, 100699. [\[CrossRef\]](#)
4. Gu, Y.; Zou, T.; Liu, H.; Lin, Y.; Ren, H.; Li, Q. Status and Challenges of Marine Current Turbines: A Global Review. *J. Mar. Sci. Eng.* **2024**, *12*, 884. [\[CrossRef\]](#)
5. Fredriksson, S.T.; Broström, G.; Bergqvist, B.; Lennblad, J.; Nilsson, H. Modelling Deep Green Tidal Power Plant Using Large Eddy Simulations and the Actuator Line Method. *Renew. Energy* **2021**, *179*, 1140–1155. [\[CrossRef\]](#)
6. Jeffcoate, P.; McDowell, J. Performance of PLAT-I, a Floating Tidal Energy Platform for Inshore Applications. In Proceedings of the 12th European Wave and Tidal Energy Conference, Cork, Ireland, 27 August–2 September 2017.
7. Dillenburg-Keenan, J.; Miller, C.; Sellar, B. On the Performance of a Horizontally Mounted ADCP in an Energetic Tidal Environment for Floating Tidal Turbine Applications. *Sensors* **2024**, *24*, 4462. [\[CrossRef\]](#) [\[PubMed\]](#)
8. Díaz-Dorado, E.; Carrillo, C.; Cidras, J.; Román, D.; Grande, J. Performance Evaluation and Modelling of the Atir Marine Current Turbine. *IET Renew. Power Gener.* **2021**, *15*, 821–838. [\[CrossRef\]](#)
9. Batten, W.M.J.; Bahaj, A.S.; Molland, A.F.; Chaplin, J.R. Hydrodynamics of Marine Current Turbines. *Renew. Energy* **2006**, *31*, 249–256. [\[CrossRef\]](#)
10. Grasso, F. Design and Optimization of Tidal Turbine Airfoil. *J. Aircr.* **2012**, *49*, 636–643. [\[CrossRef\]](#)
11. Goundar, J.N.; Ahmed, M.R. Design of a Horizontal Axis Tidal Current Turbine. *Appl. Energy* **2013**, *111*, 161–174. [\[CrossRef\]](#)
12. Jo, C.H.; Lee, K.H.; Hwang, S.J. Design Procedure to Optimize the Performance of a Tidal Current Turbine Blade. *WIT Trans. Ecol. Environ.* **2017**, *224*, 375–384. [\[CrossRef\]](#)
13. Morgado, J.; Vizinho, R.; Silvestre, M.A.R.; Páscoa, J.C. XFOIL vs CFD Performance Predictions for High Lift Low Reynolds Number Airfoils. *Aerosp. Sci. Technol.* **2016**, *52*, 207–214. [\[CrossRef\]](#)
14. Vogel, C.R.; Willden, R.H.J.; Houlsby, G.T. Blade Element Momentum Theory for a Tidal Turbine. *Ocean Eng.* **2018**, *169*, 215–226. [\[CrossRef\]](#)
15. Zhu, F.; Ding, L.; Huang, B.; Bao, M.; Liu, J. Blade Design and Optimization of a Horizontal Axis Tidal Turbine. *Ocean Eng.* **2020**, *195*, 106652. [\[CrossRef\]](#)
16. Lan, F.; Liu, J.; Zhang, F.; Wang, S.; Chi, R. Blade Profile Design and Optimization of Ocean Energy Low-Speed Water Turbine. *J. Phys. Conf. Ser.* **2021**, *1995*, 12048. [\[CrossRef\]](#)
17. El Sheshtawy, H.; el Moctar, O.; Natarajan, S. Multi-Point Shape Optimization of a Horizontal Axis Tidal Stream Turbine. *Eng* **2021**, *2*, 340–355. [\[CrossRef\]](#)

18. Zhang, Z.; Wu, B.; Liang, Y.; Tang, J.; Li, J.; Liu, W.; Chen, S.; Cui, L. Optimized Design of Tidal Current Turbine Airfoil for Water Efficiency. *Desalin. Water Treat.* **2023**, *314*, 395–405. [[CrossRef](#)]
19. Yeo, E.J.; Kennedy, D.M.; O'Rourke, F. Tidal Current Turbine Blade Optimisation with Improved Blade Element Momentum Theory and a Non-Dominated Sorting Genetic Algorithm. *Energy* **2022**, *250*, 123720. [[CrossRef](#)]
20. Wang, L.; Xu, J.; Wang, Z.; Zhang, B.; Luo, Z.; Yuan, J.; Tan, A.C.C. A Novel Cost-Efficient Deep Learning Framework for Static Fluid–Structure Interaction Analysis of Hydrofoil in Tidal Turbine Morphing Blade. *Renew. Energy* **2023**, *208*, 367–384. [[CrossRef](#)]
21. Li, C.; Liang, B.; Yuan, P.; Liu, B.; Zhao, M.; Zhang, Q.; Tan, J.; Liu, J. Tidal Turbine Blade Design Optimization Based on Coupled Deep Learning and Blade Element Momentum Theory. *Phys. Fluids* **2024**, *36*, 055110. [[CrossRef](#)]
22. Xu, J.; Wang, L.; Yuan, J.; Luo, Z.; Wang, Z.; Zhang, B.; Tan, A.C.C. DLFSI: A Deep Learning Static Fluid-Structure Interaction Model for Hydrodynamic-Structural Optimization of Composite Tidal Turbine Blade. *Renew. Energy* **2024**, *224*, 120179. [[CrossRef](#)]
23. Molland, A.F.; Bahaj, A.S.; Chaplin, J.R.; Batten, W.M.J. Measurements and Predictions of Forces, Pressures and Cavitation on 2-D Sections Suitable for Marine Current Turbines. *Proc. Inst. Mech. Eng. Part M J. Eng. Marit. Environ.* **2004**, *218*, 127–138. [[CrossRef](#)]
24. Baltazar, J.; Machado, J.; Falca o de Campos, J.A.C. Hydrodynamic Design and Analysis of Horizontal Axis Marine Current Turbines with Lifting Line and Panel Methods. In *Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering, Rotterdam, The Netherlands, 19–24 June 2011*; ASME: New York, NY, USA, 2011; Volume 44373, pp. 453–465.
25. Ge, M.; Sun, C.; Zhang, G.; Coutier-delgosha, O.; Fan, D. Combined Suppression Effects on Hydrodynamic Cavitation Performance in Venturi-Type Reactor for Process Intensification. *Ultrason. Sonochem.* **2022**, *86*, 106035. [[CrossRef](#)] [[PubMed](#)]

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.