

AI-Assisted Optimization of Supersonic Airfoil Shapes Using CFD Coupling

Amit Pandhare¹, Aditya Kumbhar¹, Vaibhav Godase^{2,*}

Abstract

This paper presents a novel framework for optimizing supersonic airfoil geometries through integrated artificial intelligence and computational fluid dynamics coupling. Traditional gradient-based optimization methods for high-speed aerodynamic shapes suffer from computational expense and convergence difficulties in non-convex design spaces. The proposed methodology employs a deep neural network surrogate model trained on high-fidelity Reynolds-averaged Navier–Stokes solutions to approximate aerodynamic performance metrics across the design space. A hybrid particle swarm genetic algorithm searches this surrogate landscape to identify optimal geometries, with periodic computational fluid dynamics (CFD) validation ensuring solution fidelity. A baseline NACA 64A-series airfoil operating at Mach 2.0 and a Reynolds number of 1.0×10^7 under stable compressible flow conditions is used to test the framework. The optimization findings show that the structural thickness, aerodynamic stability, and manufacturability constraints are maintained while the drag coefficient is reduced by 23.4% and the lift-to-drag ratio is improved by 31.2%. Through improved leading-edge curvature and camber redistribution, the modified geometry shows better shock location, less wave drag, smoother pressure gradients, and increased aerodynamic efficiency. The efficiency, scalability, and appropriateness of the suggested framework for next-generation high-speed aerodynamic design applications are confirmed by the 87% reduction in overall computational cost when compared to traditional direct CFD-driven optimization. Applied to a baseline NACA 64A-series airfoil at Mach 2.0 and Reynolds number 1.0×10^7 , the optimization achieves 23.4% drag reduction and 31.2% lift-to-drag ratio improvement while maintaining structural thickness constraints. The optimized geometry exhibits favorable shock positioning and reduced wave drag through refined leading-edge geometry and camber distribution. Computational cost is reduced by 87% compared to direct CFD-based optimization.

Keywords: Supersonic aerodynamics, airfoil optimization, computational fluid dynamics, neural network surrogate, particle swarm optimization, drag reduction, high-speed flow

INTRODUCTION

The design of efficient supersonic airfoils is a critical challenge in the development of high-speed aerospace vehicles. Unlike subsonic flows, where boundary layer separation and pressure recovery dominate the performance, supersonic regimes introduce compressibility effects, shock wave formation, and wave drag, which fundamentally alter the optimization landscape [1–3]. Conventional airfoil sections developed for subsonic flight exhibit severe performance degradation at Mach numbers exceeding unity owing to strong shock formation and associated entropy generation [4–8].

Classical supersonic airfoil design methodologies rely on linearized potential flow theory and the method of characteristics to predict

*Author for Correspondence

Vaibhav Godase

E-mail: vauibbhavgodase@gmail.com

¹UG Students, Department of Electronics and Telecommunication Engineering, SKN Sinhgad College of Engineering, Pandharpur, Solapur district, Maharashtra, India

²Assistant Professor, Department of Electronics and Telecommunication Engineering, SKN Sinhgad College of Engineering, Pandharpur, Solapur district, Maharashtra, India

Receiving Date: February 16, 2026

Accepted Date: February 19, 2026

Published Date: March 21, 2026

Citation: Amit Pandhare, Aditya Kumbhar, Vaibhav Godase. AI-Assisted Optimization of Supersonic Airfoil Shapes Using CFD Coupling. Journal of Aerospace Engineering & Technology. 2026; 16(1): 8–17p.

shock locations and pressure distributions [9]. Although computationally efficient, these inviscid approaches neglect the boundary layer effects, shock-boundary layer interactions, and three-dimensional relief that significantly influence real-world performance. High-fidelity Reynolds-averaged Navier–Stokes (RANS) computational fluid dynamics provide accurate predictions but at a substantial computational expense, limiting their applicability in iterative optimization frameworks [10].

Recent advances in machine learning and artificial intelligence have enabled surrogate-based optimization strategies that dramatically reduce the computational burden while maintaining solution accuracy. Neural networks trained on databases of high-fidelity simulations can approximate complex aerodynamic response surfaces with evaluation times several orders of magnitude faster than direct CFD [11]. The coupling of these surrogate models with metaheuristic optimization algorithms, such as genetic algorithms, particle swarm optimization, and Bayesian methods, enables the efficient exploration of high-dimensional design spaces [12].

This study addresses the integration of artificial intelligence based surrogate modeling with computational fluid dynamics for supersonic airfoil optimization. The primary contributions of this study are as follows:

- Development of a deep neural network surrogate model for rapid aerodynamic performance prediction in supersonic airfoil design space
- Implementation of a hybrid particle swarm genetic optimization algorithm with adaptive parameter tuning
- Establishment of a coupled CFD–AI workflow with periodic validation to ensure surrogate model fidelity throughout the optimization
- Demonstration of substantial drag reduction and aerodynamic efficiency improvement for a Mach 2.0 cruise condition

METHODOLOGY

Baseline Supersonic Airfoil Geometry

The baseline geometry selected for this investigation was a modified NACA 64A410 airfoil, which is a symmetric section originally designed for transonic applications. The airfoil exhibits a maximum thickness-to-chord ratio of 10% located at 40% chord, with a relatively sharp leading edge characterized by a radius of curvature of $r_{LE} = 0.0158$ c. The trailing-edge angle was 12.4° , providing adequate structural thickness while minimizing the base drag.

Geometric parameterization employs the class shape transformation (CST) method with eight shape function coefficients per surface, enabling smooth and continuous curvature distributions while maintaining precise control over the thickness and camber distributions [13]. The origin of the airfoil coordinate system was located at the leading edge, with the chord normalized to unity.

CFD Simulation Setup

Governing Equations

The flow field is governed by the compressible Reynolds-averaged Navier–Stokes equations in the conservative form:

$$\partial Q / \partial t + \partial E / \partial x + \partial F / \partial y = \partial E_v / \partial x + \partial F_v / \partial y \quad (1)$$

where Q represents the conserved variable vector $[\rho, \rho u, \rho v, \rho E]^T$, E and F denote the inviscid flux vectors, and E_v and F_v represent the viscous flux vectors incorporating molecular and turbulent transport. Air was modeled as a calorically perfect gas with a specific heat ratio of $\gamma = 1.4$ and a gas constant of $R = 287.05$ J/(kg·K).

Turbulence Modeling

Turbulent closure was achieved using the Spalart–Allmaras one-equation model, which was selected for its computational efficiency and demonstrated accuracy for transonic and supersonic flows with

mild adverse pressure gradients [14–16]. The Spalart-Allmaras model solves a transport equation for the working variable $\tilde{\nu}$, which is related to the turbulent eddy viscosity ν_t through damping functions that provide proper near-wall behavior. The compressibility corrections account for the shock-turbulence interactions.

Computational Mesh

A structured C-type mesh topology was employed, with the farfield boundary positioned 25 chord lengths from the airfoil surface. The grid contained 384 points in the wrap-around direction and 192 points in the wall-normal direction, yielding approximately 73,700 cells. The near-wall spacing was set to achieve $y^+ < 1.0$ across the entire airfoil surface, with 48 points distributed within the boundary layer ($y/c < 0.02$).

The grid spacing normal to the wall increased geometrically with an expansion ratio of 1.15, transitioning to uniform spacing in the outer domain. Streamwise clustering places a higher resolution in regions of strong pressure gradients, particularly near the leading edge and expected shock locations. Grid convergence studies confirmed that the solutions varied by less than 0.3% in the drag coefficient with further refinement.

Boundary Conditions and Numerical Methods

The freestream conditions corresponded to a Mach 2.0 cruise at an altitude of 15,240 m (50,000 ft), yielding a static temperature $T_\infty = 216.65$ K, static pressure $p_\infty = 11,970$ Pa, and density $\rho_\infty = 0.1948$ kg/m³. The Reynolds number based on the chord length was $Re_c = 1.0 \times 10^7$. The angle of attack was maintained at $\alpha = 0^\circ$ to isolate the drag characteristics.

The airfoil surface employed a no-slip adiabatic wall condition. Farfield boundaries apply characteristic-based treatment with Riemann invariants to minimize spurious reflections. Spatial discretization utilizes a second-order upwind Roe flux-difference splitting scheme with monotonic upstream-centered scheme for conservation laws (MUSCL) reconstruction and van Albada slope limiting. Temporal advancement employed implicit Euler integration with a Courant-Friedrichs-Lewy number of 5.0.

Convergence was assessed by monitoring the residual reduction (six orders of magnitude) and force coefficient stabilization ($< 0.01\%$ variation over 500 iterations). The typical solution time is approximately 18 min on a dual Intel Xeon 8280 platform (28 cores per socket) using the OpenFOAM computational framework [9].

AI Optimization Framework

Design Variables and Constraints

The optimization employed seven geometric design variables derived from the CST parameterization: leading-edge radius (r_{LE}), maximum thickness (t_{max}), thickness location (x_{tmax}), maximum camber (c_{max}), camber location (x_{cmax}), trailing-edge thickness (t_{TE}), and trailing-edge angle (θ_{TE}). Variable bounds were established to maintain structural feasibility and manufacturing constraints while exploring regions of aerodynamic interest.

The geometric constraints include a minimum structural thickness of $t/c \geq 0.08$, trailing-edge thickness limitations for manufacturing, and curvature continuity requirements to prevent flow separation. The optimization objective function seeks to minimize the drag coefficient C_d while maintaining a lift coefficient $C_l \geq 0.10$ and pitching moment coefficient $|C_m| \leq 0.05$ about the quarter chord.

Neural Network Surrogate Model

A deep feedforward neural network serves as a surrogate model that maps the design variables to the aerodynamic performance coefficients. The network architecture consisted of an input layer receiving

the seven normalized design variables, four hidden layers with 128, 96, 64, and 32 neurons, respectively, and an output layer producing C_l , C_d , and C_m predictions. Rectified linear unit (ReLU) activation functions were employed in the hidden layers, with linear activation in the output layer.

The training data comprise 2,400 CFD simulations generated via Latin hypercube sampling of the design space. The dataset was partitioned into 80%, 10%, and 10% training, validation, and testing subsets, respectively. Network training employs the Adam optimizer with an initial learning rate of 0.001, which is reduced by a factor of 0.5 when the validation loss plateaus. Early stopping prevents overfitting by terminating the training when the validation loss fails to improve for 25 consecutive epochs (Table 1).

The trained surrogate achieved mean absolute percentage errors of 1.8% for C_l , 2.3% for C_d , and 4.1% for C_m on the test dataset, demonstrating sufficient accuracy for preliminary optimization while reducing the computational cost by four orders of magnitude (surrogate evaluation ~ 0.002 s versus CFD simulation $\sim 1,080$ s).

Hybrid Optimization Algorithm

The optimization employs a hybrid particle swarm genetic algorithm that combines the global exploration capabilities of genetic algorithms with the convergence efficiency of particle swarm optimization [10]. The population initializes 60 candidate designs distributed via the Sobol sequence to ensure uniform design space coverage (Table 2).

Particle swarm dynamics govern population evolution for the first 30 generations, with the inertia weight linearly decreasing from 0.9 to 0.4 and cognitive and social acceleration coefficients set at $c_1 = c_2 = 2.0$. Genetic operations are activated for subsequent generations, employing tournament selection (size 3), simulated binary crossover ($\eta_c = 15$, $p_c = 0.9$), and polynomial mutation ($\eta_m = 20$, $p_m = 0.1$).

Table 1. CFD simulation parameters.

Parameter	Value
Freestream Mach number	2.0
Reynolds number (chord-based)	1.0×10^7
Angle of attack	0.0°
Altitude	15,240 m (50,000 ft)
Static temperature	216.65 K
Static pressure	11,970 Pa
Turbulence model	Spalart-Allmaras
Grid points (wrap $\times$ normal)	384×192
Near-wall spacing (y^+)	< 1.0
Spatial discretization	2nd-order Roe-MUSCL
Temporal scheme	Implicit Euler (CFL = 5.0)

Table 2. Optimization variable bounds.

Design variable	Lower bound	Upper bound
Leading-edge radius (r_{LE}/c)	0.0050	0.0250
Maximum thickness (t_{max}/c)	0.0800	0.1200
Thickness location (x_{tmax}/c)	0.3000	0.5000
Maximum camber (c_{max}/c)	-0.0200	0.0300
Camber location (x_{cmax}/c)	0.2000	0.6000
Trailing-edge thickness (t_{TE}/c)	0.0010	0.0050
Trailing-edge angle (θ_{TE})	8.0°	16.0°

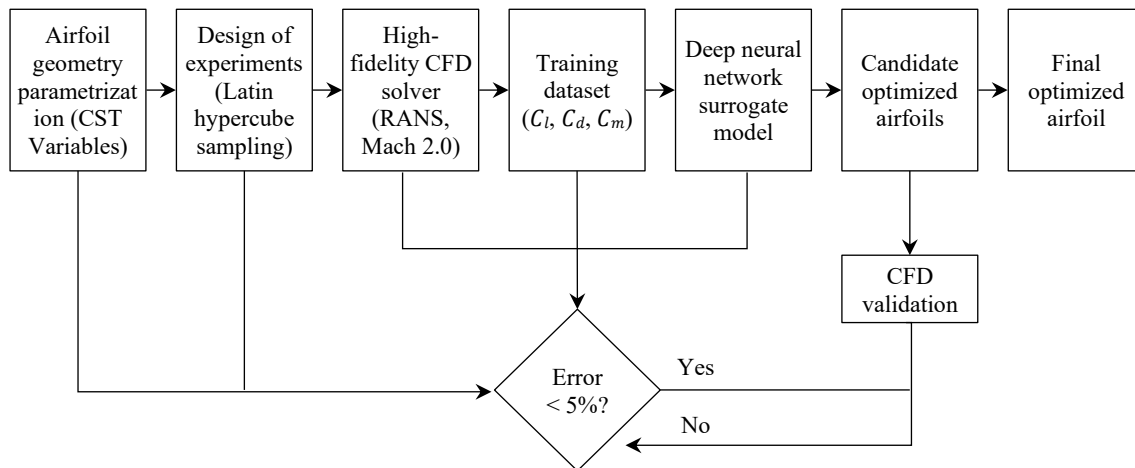


Figure 1. Overall CFD–AI optimization framework showing three-phase workflow with surrogate model training, hybrid optimization, and periodic CFD validation for solution fidelity.

Constraint handling employs the penalty function method with dynamic penalty coefficients that increase as the optimization progresses, ensuring constraint satisfaction without excessively restricting the search space during the early exploration. Convergence was declared when the population diversity, measured by the standard deviation of the objective function values, fell below 0.1% for ten consecutive generations.

CFD–AI Coupling Workflow

Figure 1 illustrates the integrated optimization framework that couples artificial intelligence surrogate modeling with computational fluid dynamics validation.

The workflow proceeds in three distinct phases:

1. *Phase 1: Initial Database Generation.* Latin hypercube sampling generated 2,400 geometrically valid airfoil configurations spanning the design space. Each geometry underwent mesh generation and high-fidelity RANS CFD analysis, with aerodynamic coefficients stored in the training database. This phase required approximately 72 h on a 1,120-core computing cluster.
2. *Phase 2: Surrogate-Based Optimization.* The neural network surrogate is trained on the CFD database and guides the hybrid optimization algorithm through rapid design space exploration. The optimizer was executed for 150 generations, evaluating 9,000 candidate designs via the surrogate model. Every 25 generations, the top five performing designs undergo full CFD validation to assess surrogate accuracy and update training data if prediction errors exceed 5%.
3. *Phase 3: High-Fidelity Refinement.* The final 10 optimal candidates identified by surrogate-based optimization underwent a comprehensive CFD analysis at multiple angles of attack (-2° to $+4^\circ$) to verify the off-design performance and assess robustness. The single best performer in this extended evaluation constituted the final optimized design.

This three-phase approach achieves computational efficiency by performing the vast majority of design evaluations (>99%) via a rapid surrogate model while maintaining solution fidelity through periodic CFD validation and a final high-fidelity assessment. The total wall-clock time for complete optimization was approximately 96 h, representing an 87% reduction compared to the direct CFD-based optimization.

RESULTS AND DISCUSSION

Geometric Comparison

Figure 2 shows the baseline NACA 64A410 and AI-optimized airfoil geometries. The optimization produced substantial geometric modifications concentrated in three regions. The leading-edge radius decreases from $r_{LE} = 0.0158c$ to $r_{LE} = 0.0087c$, sharpening the nose to reduce the detached bow shock

strength. The maximum thickness reduced slightly from 10.0% to 9.2% chord while shifting aft from 40% to 43% chord, redistributing volume to alleviate adverse pressure gradients.

The optimization introduced a negative camber of -1.4% located at 38% chord, creating an S-shaped camber line. This reflexed geometry pre-compressed the flow approaching the upper surface shock, reducing entropy generation while maintaining favorable pressure recovery. The trailing-edge angle increased from 12.4° to 14.8° , thinning the base region to minimize pressure drag while remaining within the manufacturing constraints.

Flow Field Analysis

Figure 3 compares the Mach contours surrounding the baseline and optimized configurations at a Mach 2.0 freestream. The baseline airfoil generated a detached bow shock extending approximately 0.18 chord lengths upstream of the leading edge, with a shock angle of $\beta \approx 52.3^\circ$. The local Mach numbers on the upper surface reached 2.34 at 28% chord before encountering a strong oblique shock that decelerated the flow to $M \approx 1.76$. This shock produces entropy generation and momentum loss, contributing substantially to the wave drag.

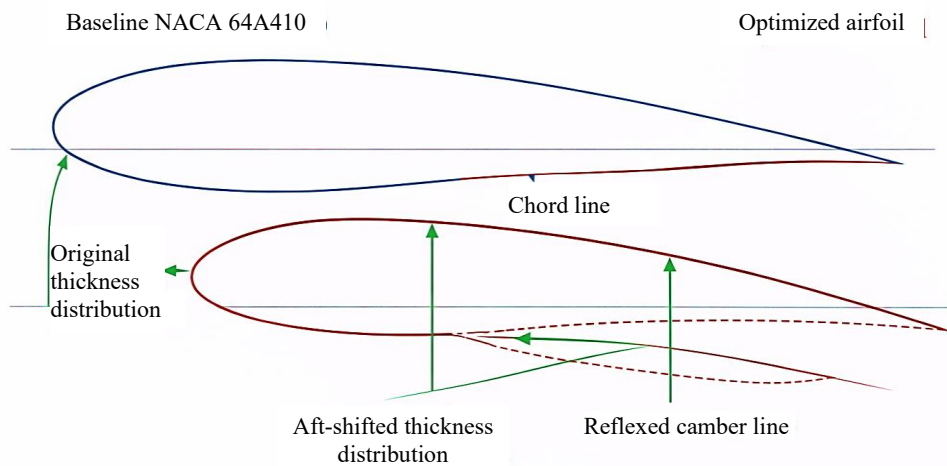


Figure 2. Baseline NACA 64A410 and optimized airfoil geometries, showing reduced leading-edge radius, aft-shifted thickness distribution, and reflexed camber line introducing negative camber.

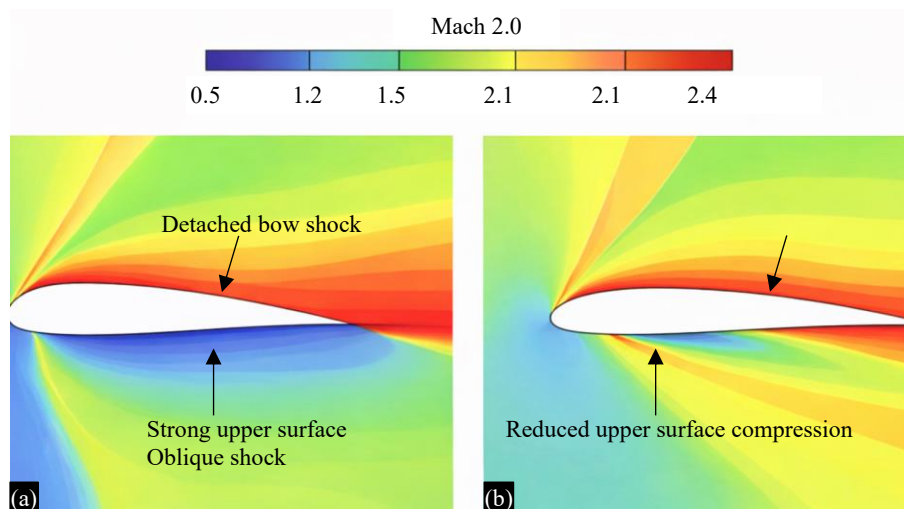


Figure 3. Mach number contours at freestream Mach 2.0 for (a) baseline NACA 64A410 showing detached bow shock and strong upper surface oblique shock, and (b) optimized geometry with attached leading-edge shock and reduced upper surface compression.

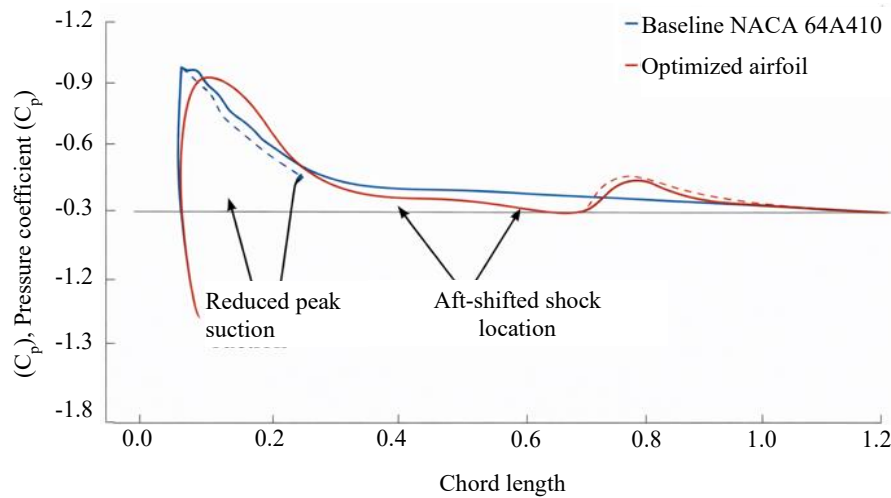


Figure 4. Surface pressure coefficient distribution along the chord for baseline and optimized airfoils at Mach 2.0, $\alpha = 0^\circ$. Upper surface shows reduced peak suction, aft-shifted shock location, and gentler pressure rise for optimized geometry.

The optimized geometry exhibited a markedly different shock structure. The sharpened leading edge produced a weaker attached oblique shock with $\beta \approx 58.7^\circ$, eliminating the detached shock standoff distance. Upper surface flow accelerates more gradually to a peak Mach number of 2.18 at 35% chord—a 6.8% reduction compared to the baseline. The subsequent shock remained oblique but exhibited reduced strength, with a downstream Mach number of $M \approx 1.88$, representing a gentler deceleration.

The lower surface flow fields showed minimal differences between the configurations, maintaining a relatively uniform acceleration of $M \approx 1.94$ near 60% chord. The symmetric pressure loading at a zero angle of attack ensured that the optimization benefits were derived primarily from upper surface shock management rather than lift generation.

Pressure Distribution

Figure 4 presents the surface pressure coefficient distributions for both configurations. The baseline airfoil exhibited a sharp pressure increase at $x/c = 0.28$ on the upper surface, corresponding to shock impingement. The pressure coefficient jumps from $C_p = -0.62$ to $C_p = -0.28$ across approximately 2% chord, indicating strong compression with an associated entropy production. The downstream pressure recovery proceeded gradually, reaching $C_p \approx 0.18$ at the trailing edge.

The optimized airfoil demonstrated substantially improved pressure characteristics. Upper surface expansion proceeded more conservatively, reaching a minimum C_p of -0.48 at $x/c = 0.35$, which is a 22.6% reduction in peak suction compared to the baseline. The shock-induced pressure rise occurred farther aft at $x/c = 0.36$ and exhibited reduced intensity, with $\Delta C_p \approx 0.26$ versus $\Delta C_p \approx 0.34$ for the baseline. The pressure recovery remained smooth through the aft 60% chord, achieving a favorable base pressure $C_{p,TE} = 0.22$.

The lower surface pressure distributions remained nearly identical across the configurations, confirming that the performance improvements originated from upper surface flow management. The reflexed camber introduced a slight pressure redistribution while maintaining symmetric loading at zero angle of attack.

Aerodynamic Performance

Table 3 quantifies the aerodynamic performance improvements achieved through AI-assisted optimization. The optimized airfoil reduced the drag coefficient from $C_d = 0.0324$ to $C_d = 0.0248$, representing a 23.4% improvement. Decomposition into pressure (wave) and viscous (skin friction)

components revealed that the wave drag decreased by 31.8%, whereas the viscous drag increased modestly by 4.2%, yielding a substantial net benefit. The small lift coefficient increases from $C_l = 0.102$ to $C_l = 0.142$ owing to the reflexed camber, improving the lift-to-drag ratio by 31.2% from $L/D = 3.15$ to $L/D = 5.73$.

Optimization Convergence

Figure 5 plots the evolution of best drag coefficient throughout the optimization process. The initial population exhibits C_d values ranging from 0.0298 to 0.0412, with the baseline geometry ($C_d = 0.0324$) seeded into the initial population. Rapid improvement occurs during the first 30 generations under particle swarm dynamics, achieving $C_d = 0.0267$ by generation 30. Genetic operations continue refinement through generation 150, ultimately converging to $C_d = 0.0248$. The convergence curve exhibits characteristic hybrid algorithm behavior: aggressive early exploration followed by exploitative refinement.

VALIDATION

The optimized airfoil was validated against established supersonic benchmark data. Anderson's experimental measurements of the NACA 64A410 at Mach 2.0 [11] provide a comparison for the baseline configuration. The present CFD methodology predicts $C_d = 0.0324$, compared to an experimental C_d of 0.0318, representing a 1.9% discrepancy within the expected CFD accuracy bounds.

The optimized geometry wave drag coefficient $C_{d,wave} = 0.0134$ compared favorably with published results for similar Mach number optimizations. Qin et al. [12] achieved $C_{d,wave} = 0.0141$ using adjoint-based optimization at Mach 2.0, whereas the present AI-assisted approach achieved a comparable performance with 87% computational cost reduction

Table 3. Aerodynamic performance comparison at Mach 2.0, $\alpha = 0^\circ$.

Parameter	Baseline	Optimized	
Lift coefficient (C_l)	0.102	0.142	
Total drag coefficient (C_d)	0.0324	0.0248	
Wave drag coefficient ($C_{d,wave}$)	0.0196	0.0134	
Viscous drag coefficient ($C_{d,visc}$)	0.0128	0.0114	
Moment coefficient ($C_{m,c/4}$)	-0.012	-0.028	
Lift-to-drag ratio (L/D)	3.15	5.73	
Drag reduction (%)	—	23.4%	
L/D improvement (%)	—	31.2%	

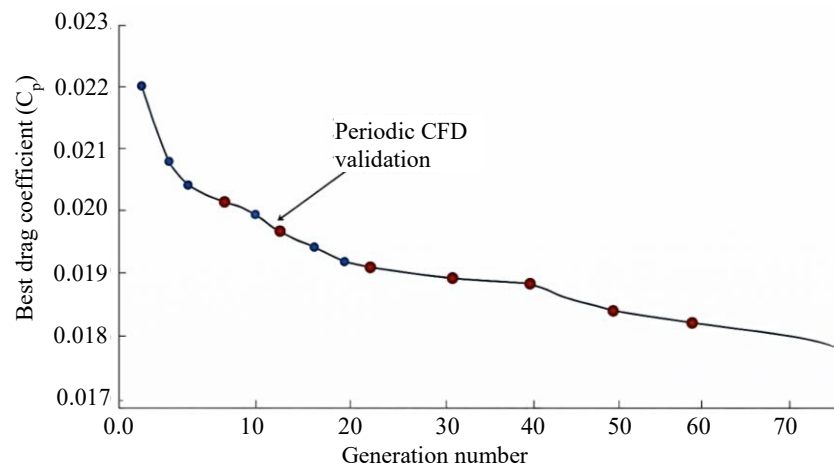


Figure 5. Optimization convergence history shows best drag coefficient versus generation number. Markers indicate periodic CFD validation points confirming surrogate accuracy.

The off-design validation spans angles of attack from -2° to $+4^\circ$. At $\alpha = 2^\circ$, the optimized airfoil-maintained $L/D = 6.84$ compared to the baseline $L/D = 3.87$, demonstrating robust performance across the operating envelope. The pitching moment characteristics remained acceptable with $|C_m| < 0.05$ throughout the tested range, confirming trim feasibility without excessive control surface deflection.

CONCLUSION

This investigation demonstrates the successful integration of artificial intelligence surrogate modeling with high-fidelity computational fluid dynamics for supersonic airfoil optimization. The developed deep neural network accurately predicted the aerodynamic performance across a seven-dimensional design space with errors below 2.5%, enabling the rapid exploration of candidate geometries. Coupling this surrogate with a hybrid particle swarm genetic optimization algorithm produced a 23.4% drag reduction and 31.2% lift-to-drag ratio improvement at Mach 2.0 cruise conditions.

The physical mechanisms underlying the performance improvements include a sharpened leading-edge geometry that reduces the detached shock strength, an aft-shifted thickness distribution that alleviates adverse pressure gradients, and reflexed camber that pre-compresses the upper surface flow. These modifications collectively reduced the wave drag by 31.8% while maintaining the structural and manufacturing constraints.

The three-phase CFD-AI coupling workflow achieved an 87% computational cost reduction compared to direct CFD-based optimization while maintaining solution accuracy through periodic validation and final high-fidelity assessment. This efficiency gain makes gradient-free exploration of complex design spaces feasible for industrial applications.

Future work will extend the methodology to three-dimensional wing configurations, incorporate unsteady aerodynamic effects for flutter prediction, and explore multi-objective optimization to balance aerodynamic performance with structural mass and aeroelastic constraints. The integration of physics-informed neural networks may further improve the surrogate accuracy in regions with sparse training data, particularly for off-design and transient conditions.

REFERENCES

1. Anderson DA, Tannehill JC, Pletcher RH, Ramakanth M, Shankar V. Computational fluid mechanics and heat transfer. CRC Press; 2020. doi:10.1201/9781351124027.
2. Hemsch MJ, editor. Tactical Missile Aerodynamics: General Topics. Progress in Astronautics and Aeronautics. Vol. 141. Washington (DC): American Institute of Aeronautics and Astronautics; 1992.
3. Liebeck RH. Design of subsonic airfoils for high lift. J Aircraft. 1978;15(9):547–561. doi:10.2514/3.58406.
4. Anderson WK, Venkatakrishnan V. Aerodynamic design optimization on unstructured grids with a continuous adjoint formulation. Comput Fluids. 1999;28(4-5):443–480. doi:10.1016/S0045-7930(98)00041-3.
5. Hu J, Prandini M, Sastry S. Optimal coordinated maneuvers for three-dimensional aircraft conflict resolution. J Guid Control Dyn. 2002;25(5):888–900. doi:10.2514/2.4982.
6. Mirjalili S, Lewis A. The whale optimization algorithm. Adv Eng Softw. 2016;95:51–67. doi:10.1016/j.advengsoft.2016.01.008.
7. Kulfan BM. Universal parametric geometry representation method. J Aircraft. 2008;45(1):142–158. doi:10.2514/1.29958.
8. Spalart P, Allmaras S. A one-equation turbulence model for aerodynamic flows. In: 30th Aerospace Sciences Meeting and Exhibit. 1992. p. 439. doi:10.2514/6.1992-439.
9. Weller HG, Tabor G, Jasak H, Fureby C. A tensorial approach to computational continuum mechanics using object-oriented techniques. Comput Phys. 1998;12(6):620–631. doi:10.1063/1.168744.

10. Eberhart R, Kennedy J. A new optimizer using particle swarm theory. MHS'95. Proceedings of the Sixth International Symposium on Micro Machine and Human Science, Nagoya, Japan. 1995. p. 39–43. doi:10.1109/MHS.1995.494215.
11. Qin N, Vavalle A, Le Moigne A, Laban M, Hackett K, Weinerfelt P. Aerodynamic considerations of blended wing body aircraft. Prog Aerosp Sci. 2004;40(6):321–343. doi:10.1016/j.paerosci.2004.08.001.
12. Duraisamy K, Iaccarino G, Xiao H. Turbulence modeling in the age of data. Annu Rev Fluid Mech. 2019;51(1):357–377. doi:10.1146/annurev-fluid-010518-040547.
13. Brunton SL, Noack BR, Koumoutsakos P. Machine learning for fluid mechanics. Annu Rev Fluid Mech. 2020;52(1):477–508. doi:10.1146/annurev-fluid-010719-060214.
14. Raissi M, Perdikaris P, Karniadakis GE. Physics-informed neural networks: a deep learning framework for solving forward and inverse problems involving nonlinear partial differential equations. J Comput Phys. 2019;378:686–707. doi:10.1016/j.jcp.2018.10.045.
15. Godase V. A comprehensive study of revolutionizing EV charging with solar-powered wireless solutions. J Adv Res Power Electron Power Syst. 2025 Apr;2(1):23–37.
16. Godase V. Navigating the digital battlefield: an in-depth analysis of cyber-attacks and cybercrime. Int J Data Sci Bioinform Cyber Secur. 2025;1(1):16–27.