Progress of CFD-based Aerodynamics Optimization

Jinhua Cheng*

Department of Physical Sciences, University of California, Irvine, 92697, USA

*Corresponding author: jinhuc2@uci.edu

Abstract. Computational Fluid Dynamics (CFD) has become a cornerstone of modern aerodynamic optimization, enabling precise simulation and analysis of fluid flow across complex biomimetic geometries. This paper reviews the latest progress in CFD-based aerodynamic design, focusing on bionic flow control mechanisms and multi-scale optimization strategies. Inspired by natural organisms—such as the drag-reducing riblets of sharks, the stall-delaying tubercles of humpback whales, and the noise-suppressing feathers of owls—researchers have successfully incorporated biological principles into engineering applications in the aerospace and automotive fields. The review highlights three critical aspects: CFD simulation of boundary-layer interactions and vortex dynamics, fluid–structure interaction (FSI) for flexible aerodynamic designs, and AI/ML-assisted CFD acceleration for data-driven optimization. Moreover, multi-objective algorithms and surrogate modeling have significantly improved efficiency in exploring complex design spaces. The integration of CFD with artificial intelligence is paving the way toward real-time, intelligent aerodynamic design and system-level optimization. Future trends include refined multi-scale modeling, stronger coupling of physical and data-driven methods, and broader application of bionic concepts in sustainable engineering.

Keywords: Computational Fluid Dynamics (CFD); Aerodynamic Optimization; Biomimetic Design; Flow Control; Machine Learning.

1. Introduction

The significance of shape optimization has been fully demonstrated. The existing literature indicates that the shape and geometry of the re-entry vehicle shell design are crucial factors affecting vehicle performance. Continuous modeling, building, and computational simulation could effectively seek the influence of shape [1]. Similarly, in aerodynamic optimization of a bionic shell, geometric design details, such as raised structures and inclination angles, can ultimately determine the vehicle's motion performance.

Nowadays, technological advances have given us more opportunities to learn from nature. CFD is a clear example of this; it is widely used in biomimetic shell research because it enables the simulation of nanoscale structures. Meanwhile, it has an apparent advantage in visualization, adjustable parameters, and low cost [2]. CFD mainly relies on the Navier-Stokes equations, combining multiple simulation methods such as Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) to solve problems flexibly [3].

This paper aims to review the progress of CFD application in biomimetic fields and focus on bionic drag reduction and flow control. At the same time, it reveals challenges in this domain and looks forward to further prospects.

2. Inspiration from Natural Shells in Flow Field Control

In recent years, natural creatures have inspired engineers by their unique counteracting resistance strategies. Existing studies focus on model building to achieve optimization, further exploiting the influence of specific features on aerodynamic properties.

2.1. Aquatic Animal

Aquatic creatures offer inspiration for drag reduction and flow control through their surface structures and shapes. For the first time, Walsh and Linemann verified that the riblets on shark skin can effectively reduce frictional drag on a turbulent wall, achieving drag reductions of 5% to 8% [4]. Subsequently, in 1997, Bechert et al. optimized the experiment by adjusting the ribs, thereby improving the drag-reduction rate to 9.9% and setting a new record [5]. This study is highly cited and has become a cornerstone of drag-reduction research. In terms of Bionic lift control, Fish observed the unique tubercle structure of the humpback whale's fin on the leading edge and speculated that it could enhance lift [6]. Additionally, aquatic animals inspired the control of flow separation. One typical case is the tubercles on the leading edges of the humpback whale's pectoral fins. The leading edge of the pectoral fin of the humpback whale is not a smooth straight line, but instead a series of raised nodules. Wind-tunnel experiments and numerical studies show that this leading-edge structure delays the onset of stall at large attack angles. The mechanism is that the nodules at the leading edge generate a series of vortices distributed along the span direction, which increase the energy near the wing surface and delay the separation of the boundary layer [7].

Recently, researchers have focused on active and passive control and design optimization. For example, Guo et al. analyzed the mechanism of the bionic shark skin in non-steady separation flow by dynamic simulation. They found that the nanoscale structure could delay flow separation [7]. In summary, the bionic design derived from aquatic creatures such as sharks and whales has been effective at reducing drag and delaying stall.

2.2. Animal in the Air

Flying organisms (birds, bats, insects) provide insights into aerodynamic properties and flow control through their wing structures and flight mechanisms. Buckholz studied the wavy folds in the cross-sections of insect wings for their aerodynamic properties and explained that the wavy structure could delay stall and reduce resistance at low Reynolds numbers [8]. Spillman was inspired by the wingtip bifurcation feathers of raptors and designed wingtip appendages. The experiment shows that multiple bifurcation wingtips can reduce induced drag, providing a new appendage design for modern aircraft [9].

Carruthers et al. discovered, using high-speed cameras, that when eagles perform high-angle maneuvers, the primary feathers on the leading edge and upper surface of their wings are passively raised, forming leading-edge flaps and acting as vortex generators, expanding the angle-of-attack range before the wing stalls. This mechanism demonstrates that the bird wing contains passive flow-control devices, thereby inspiring the design of biomimetic passive flaps [10]. Additionally, Jiakun Han summarized the outstanding aerodynamic performance of the bionic dragonfly wave-shaped wing in micro aircraft. He highlighted the potential to enhance the efficiency of micro unmanned aircraft through biomimetic design [11].

The owl is a crucial subject in the study of noise reduction. It relies on three types of structures: leading-edge serrations help reduce lift fluctuations and vortex-induced noise; a downy-feather-like porous coating can dissipate small-scale pressure pulsations; and fringes or porous, compliant trailing edges can significantly weaken classical trailing-edge noise. Transplant those features into drones, Baldes, fans, and wind turbine blades could typically achieve broadband noise reduction [12].

3. Key Issue in CFD Study

To infuse biomimetic concepts into engineering design, several key problems in CFD modeling and analysis must be addressed. This section will discuss three of them: the effect of surface texture on the boundary layer, flow separation and vortex shedding, and the application of fluid-solid coupling analysis in aerodynamic optimization.

3.1. The Effect of the Surface Texture on the Boundary Layer

Utilizing CFD can provide a deep analysis of how a biomimetic microstructure could influence the boundary layer and friction drag. Choi, through Direct numerical simulation, achieved the first high-resolution capture of the idealized micro-rib drag-reduction mechanism. The simulation result shows that the micro-slots along the flow direction restrain the lateral movement of the near-wall vortices, thereby reducing turbulent friction [12]. Such a CFD study forms the foundation of the subsequent optimized design of bionic surfaces. Key recent progress includes the simulation of complex biomimetic surfaces; for example, when et al. simulated a three-dimensional array of shark skin teeth, revealing that the teeth would induce vortices at certain attack angles, delaying flow separation [13]. In general, this type of biomimetic surface texture often involves multiscale flows and the instability of the turbulent boundary layer, which can only be captured by high-precision CFD simulations and advanced turbulence models. This poses challenges for the computational sources and modeling methods; it is also a technological difficulty that needs to be overcome in the future.

3.2. Control Strategies for Flow Separation and Vortex Shedding

Flow separation can result in lift loss and resistance increase. It may also trigger vortex shedding. How to control flow control and vortex shedding is one of the crucial topics in aerodynamics optimization. CFD plays a core role in such a study: on the one hand, it can simulate the critical conditions and characteristics of flow separation at different angles and Reynolds numbers. For example, numerous studies on car tail models show that adding small appendages at the edge of the car tail or applying active suction and blowing can significantly delay wake flow separation, reduce the intensity of the wake vortex, and lower the base pressure and drag [14]. Detailed CFD data can identify the separation flow area, estimate the influence of control measures on the pressure distribution and vortex structure, and thus evaluate the control effect and guide vehicle design.

3.3. Application of Fluid-Solid Coupling (FSI) in Aerodynamic Optimization

Numerous biomimetic structures are flexible or deformable (such as the elastic bending of bird wings and the oscillation of fish fins). In engineering design, structural deformation can affect a structure's aerodynamic properties. For instance, the wings of high-speed aircraft can deform under aerodynamic forces to alter the effective angle of attack and load distribution; another example is that wind turbine blades can deflect during strong winds to prevent stall. Therefore, coupling CFD with structural mechanics for aerodynamic and elastic analysis enables accurate estimation of actual working conditions and can be used to optimize the design [15]. Apart from aircraft, the body shells of racing cars and high-speed trains also undergo deformation under aerodynamic forces. Using FSI analysis, this paper can estimate the influence of such deformation on the airflow and stability, thereby guiding the body design. In CFD, fluid-solid coupling increases simulation complexity and computational cost, requiring the simultaneous solution of the fluid and structural domains. Recently, standard methods include loosely coupled iterative methods and strong-coupling simultaneous solution methods, etc. In the field of bionics, researchers use FSI to simulate the motion of animal wings and fish fins and draw insights for optimizing the flexible-rigid structure. In general, fluid-solid coupling analysis enables collaborative optimization of the shell's aerodynamic and structural properties, advancing the translation of bionic design into engineering applications [16].

4. Multiscale Modeling and Optimization Methods

Aerodynamic optimization of Bionic shell design generally involves multiple scales (from microscopic textures to macroscopic shapes) and balances various goals, such as lift, drag, and noise. Therefore, this paper needs to develop corresponding multi-scale simulations and multi-objective optimization, combining CFD and modern algorithm technologies to boost design efficiency.

4.1. Microscopic and Macroscopic Shape Collaborative Modeling

In biomimetic design, one challenging topic is simultaneously optimizing the microstructural surface and macro shape to improve aerodynamics. Traditionally, industry design usually disposes of them separately, first determining the macrostructure, then applying the microstructure locally (or vice versa). However, natural creatures(like sharks and butterflies) offer examples of a combination of surface texture and integral shape. Hence, scholars start to attempt collaborative optimization strategies. However, because the microscopic structure is much smaller than the whole body, using CFD with the same grid will incur significant costs. One viable approach is to adopt multi-level modeling or multi-fidelity simulation: for example, using PANS simulation for the macro-geometry and LES or DNS for the local micro-structure to obtain high-precision information, and then feeding back the micro-structure effects to the macro-simulation through equivalent boundary conditions or coarsening models. Some scholars have also proposed building surrogate models using statistical or machine learning methods to learn the mapping between microstructural geometric parameters and macroaerodynamic performance, and to estimate this relationship during optimization iterations quickly [17].

4.2. Multi-objective Optimization Algorithm with CFD

Vehicle aerodynamic design involves multiple objective trade-offs, such as reducing drag while ensuring enough lift and stability. Additionally, it may include noise and heat dissipation. For such complex problems, global optimization methods such as evolutionary algorithms have demonstrated their advantages. For example, Genetic Algorithms (GA) and Particle Swarm Optimization (PSO) have been widely employed in wing/vehicle design. That algorithm can generate a series of Pareto-optimal solutions in a single search, enabling designers to choose among different trade-off scenarios [18]. In terms of implementation, CFD simulation is typically integrated into an optimization loop to evaluate the aerodynamic properties of each design. Although it can deliver high-fidelity results, the computational cost is exceptionally high (each CFD solution is time-consuming). To enhance optimization efficiency, multi-fidelity optimization and surrogate model-assisted optimization have emerged in recent years. For example, researchers infuse a surrogate model using deep learning with a genetic algorithm to present inverse design of the winglet pressure distribution. Only the aerodynamic cost function predicted by the surrogate model converges, and the optimal individual is then further verified using CFD, significantly reducing the number of CFD operations [19]. With improvements in computing power, it is possible to directly use local high-precision simulations (e.g., coupling LES or solving the NS equation) to evaluate the design and ensure reliable results. In general, the combination of multi-objective evolutionary algorithms and CFD makes automated aerodynamic optimization a reality, enabling efficient search over vast design spaces and yielding a series of excellent solutions after trade-offs.

4.3. AI/ML-based CFD Acceleration and Data-driven Optimization

In recent years, artificial intelligence (AI) and machine learning (ML) have rapidly integrated into CFD. Their application involves accelerating CFD computations, assisting in result analysis, and directly participating in optimization design, among several aspects. For example, generative adversarial networks (GANs) or autoencoders can be used to quickly refine high-resolution flow field distributions based on rough meshes or preliminary results, thereby reducing the iterative solution time; convolutional neural networks (CNNs) are used to directly predict the lift and drag coefficients of steady flow around objects based on geometric parameters, with accuracy approaching RANS calculations but much less time-consuming. On the other hand, ML is also used to improve turbulence models and enhance simulation accuracy. For instance, ML-based turbulence model corrections have become a hot topic: researchers train models on large amounts of high-fidelity data to introduce data-driven correction terms or optimize turbulence model coefficients in the RANS equations, thereby significantly improving RANS predictions of complex flows. Additionally, in design optimization, active learning algorithms can intelligently select sampling points for CFD calculations to train more accurate proxy models and guide the next design iteration. Research has combined Bayesian optimization with CFD to significantly reduce the number of computational samples in the multi-objective optimization of unmanned aircraft wing designs. In the case analysis stage, interpretability tools in ML (such as SHAP values) have also been introduced into the CFD field to analyze the influence of input geometries on results (e.g., lift and drag) and provide engineers with physical explanations. Overall, the integration of AI/ML is enabling CFD to move from "time-consuming simulations" to "real-time forecasts" and "intelligent design", allowing data-driven optimization to shine. This trend will significantly enhance the efficiency of aerodynamic design simulations, enabling engineers to screen and refine nature-inspired solutions quickly [20,21].

5. Typical Achievements and Application Scenarios

5.1. Aerospace Field: Optimization of the Shape of Aerial Vehicles and Small Aircraft

The bionic concept shows unique advantages in small-scale aircraft. For instance, researchers have applied the anterior notches of the humpback whale's fin to the wing design of small unmanned aerial vehicles (UAVs). CFD studies have shown that UAV wings with sinusoidal wave-like leading-edge undulations can significantly delay stall, increase lift, and have more stable aerodynamic responses in gusts. This notched structure is equivalent to passively generating a series of flow vortices, enhancing the wing's lift robustness in turbulent incoming flow, and is particularly suitable for small-aspect-ratio UAVs' flight in complex atmospheres. Additionally, bionics also has direct applications in micro aircraft: flapping-wing aircraft are inspired by insects and adopt a four-wing layout and flexible wing membrane design. For example, the research team developed a four-wing microplane that simulates the coordinated flapping of two pairs of wings in a dragonfly, and its stable leading-edge vortex generation mechanism and phase-control strategy are derived from CFD analysis of insect flight. Another example is that a scholar designed a retractable, small leading-edge wing based on bird wings for the front edge of a small UAV, and test results showed it effectively increases lift and stall margin at low speeds and high attack angles. In the field of high-speed aircraft, NASA and other institutions also pay attention to bionic concepts, such as researching the nose shape of the falcon (falcon) to reduce the wave resistance of high-speed warheads, or designing an active deforming wing based on the active deformation wing of the flying fish to optimize high maneuvering flight performance. These explorations all use CFD methods to evaluate the feasibility of concepts and guide prototype design. It can be predicted that, as materials and control technologies develop, more bio-inspired structures (such as retractable wings, bionic bird feather skins, etc.) will be applied to unmanned aerial vehicles and new concept aircraft to achieve more efficient aerodynamic layouts [22,23].

5.2. Automobile Field: Aerodynamic Shell Design

Aerodynamic optimization based on Computational Fluid Dynamics (CFD) has emerged as a core approach in automotive aerodynamic design. Through the integration of high-fidelity simulations, surrogate modeling, and optimization algorithms, vehicle exterior design has transitioned from an experience-driven process to an intelligent, data-guided, and physically constrained optimization. In electric vehicle design, CFD optimization is primarily used to reduce aerodynamic drag, control wake separation, and enhance cruising performance. Tran et al. proposed an aerodynamically-guided machine learning optimization framework that couples a deep learning surrogate model with CFD simulations. This framework can rapidly predict aerodynamic responses to shape changes, enabling efficient shape optimization. Research results indicate that this method can significantly reduce the drag coefficient while maintaining styling and structural constraints, enabling the vehicle to maintain a more stable wake structure and a more even lateral force distribution at high speeds. Such research demonstrates the immense potential of combining CFD and artificial intelligence for the aerodynamic design of electric vehicles [24].

6. Conclusion

Research on the flow mechanisms and aerodynamic optimization of bionic shells using Computational Fluid Dynamics (CFD) is an interdisciplinary frontier field that integrates knowledge from fluid mechanics, bionics, and computational science. As shown in the above review, natural organisms offer a wide variety of flow-field control strategies. For example, shark skin reduces frictional drag, whale fin tubercles delay stall, the alula of birds enhances lift and prevents stall, and the corrugations on insect wings optimize aerodynamics at low Reynolds numbers. With the support of CFD technology, these inspirations have been quantitatively analyzed for their mechanisms and applied to the optimization of engineering shapes, yielding a series of remarkable achievements across fields such as aerospace, rail transit, and automotive engineering. High-fidelity CFD simulations not only reveal the fluid physics behind bionic designs, enabling us to understand the efficient natural mechanisms better, but also, with the help of optimization algorithms and artificial intelligence, significantly accelerate the translation of bionic concepts from inspiration into design solutions. Looking ahead, as computing power improves and multi-scale modeling methods are refined, this paper expects to apply CFD to more complex bionic structures, achieving overall optimization from macroscopic shapes to microscopic textures and obtaining unprecedented performance improvements. Moreover, multi-objective and multidisciplinary optimization frameworks will enable bionic designs to meet various requirements, such as aerodynamic performance, structural strength, and noise control, thereby achieving system-level optimization similar to that in nature. It is foreseeable that the in-depth integration of bionics and CFD will continue to expand the boundaries of engineering design, providing continuous inspiration and technical support for the development of a new generation of efficient and green vehicles. This is not only a tribute and application of natural wisdom but also a driving force for engineering technology to develop towards greater efficiency and sustainability.

References

[1] Bechert D W, Bruse M, Hage W, Van der Hoeven J G T, Hoppe G. Experiments on drag-reducing surfaces and their optimization with adjustable geometry. Journal of Fluid Mechanics, 1997, 338: 59–87.

[2] Bounds C P, 2024. Enhancing CFD predictions with explainable machine learning for aerodynamic characteristics of idealized ground vehicles. Aerospace Science and Technology, 149: 109986.

[3] Caron C, Roy N, Lecocq G, et al. Machine learning to speed up computational fluid dynamics engineering simulations for built environments: a review. Building and Environment, 2024, 255: 111089.

[4] Carruthers A C, Thomas A L R, Taylor G K. Automatic aeroelastic devices in the wings of a steppe eagle (Aquila nipalensis). Journal of Experimental Biology, 2007, 210(23): 4136–4149.

[5] Choi H, Moin P, Kim J. Direct numerical simulation of turbulent flow over riblets. Journal of Fluid Mechanics, 2006, 177: 133–166.

[6] Dussauge T P. A reinforcement learning approach to airfoil shape optimization. Aerospace Science and Technology, 2023, 142: 108423.

[7] Fish F E, Battle J M. The tubercles on humpback whales' flippers: application of bio-inspired technology. Integrative and Comparative Biology, 2011, 51(1): 203–213.

[8] Han J, Zhang Y, Chen R, et al. Review on bio-inspired flight systems and bionic aerodynamics. Progress in Aerospace Sciences, 2020, 116: 100617.

[9] Hou D, Zhang Z, Gao Y, et al. Aerodynamic effects of time-varying corrugations on dragonfly wings in flapping flight. Bioinspiration & Biomimetics, 2024, 19(3): 036012.

[10] Jaworski J W, Peake N. Aeroacoustics of silent owl flight. Annual Review of Fluid Mechanics, 2020, 52: 395–420.

[11] Ke W, Wang D, Zhao Q, et al. Influence of leading-edge tubercles on the aerodynamic performance of a horizontal-axis wind turbine: a numerical study. Renewable Energy, 2021, 170: 1031–1042.

[12] Li W, Zhang J, Chen Y, et al. Biomimetic shark skin: design, fabrication and hydrodynamic function. Bioinspiration & Biomimetics, 2014, 9(4): 046018.

[13] Othman A K, Abdelaziz M, Ahmed M R. Aerial and aquatic biological and bioinspired flow control strategies. Progress in Aerospace Sciences, 2023, 146: 100927.

[14] Poloni C, Onesti L, Giurgevich A, et al. Multi-objective aerodynamic optimization by means of robust and efficient genetic algorithms. Structural and Multidisciplinary Optimization, 2004, 27(1–2): 64–74.

[15] Shirvani A, Yaghoubi M, Hashemi M R. A deep learning–genetic algorithm approach for aerodynamic inverse design via optimization of pressure distribution. Aerospace Science and Technology, 2024, 150: 110021.

[16] Spillman J J. The use of wing tip sails to reduce vortex drag. Aeronautical Journal, 2016, 120(1226): 1451–1465.

[17] Sudhakar S, Balakrishnan S. Influence of leading-edge tubercles on aerodynamic characteristics of a high aspect-ratio UAV. Aerospace Science and Technology, 2017, 66: 86–97.

[18] Sudin M N, Abdullah M Z, Ariffin A K, et al. Review of research on vehicle aerodynamic drag reduction methods. Applied Mechanics and Materials, 2014, 663: 153–158.

[19] Tran J, Lee K, Cho Y, et al. Aerodynamics-guided machine learning for design optimization of electric vehicles. Energy Reports, 2024, 10: 1090–1102.

[20] Wang L, Zhang C, Gao F, et al. A relay-based probabilistic prediction model for multi-fidelity scenarios in the total pressure loss of a compressor cascade with micro-textured surfaces. Energy Conversion and Management, 2025, 316: 118758.

[21] Walsh M J, Lindemann A M. Optimization and application of riblets for turbulent drag reduction. AIAA Journal, 1984, 22(4): 485–489.

[22] Zhan Z, Liu J, Zhang X, et al. Review of computational fluid dynamics analysis in biomimetic applications for underwater vehicles. Ocean Engineering, 2024, 298: 117463.

[23] Zhou E, Basri E this paper, Abdullah M F, et al. Computational fluid dynamics analysis in biomimetics applications: a review from aerospace engineering perspective. Aerospace Science and Technology, 2023, 137: 107874.