fluids

MDPI

Editorial

Recent Advances in Fluid Mechanics: Feature Papers, 2024

Giuliano De Stefano [1,*] and D. Andrew S. Rees [2]

[1] Engineering Department, University of Campania Luigi Vanvitelli, Via Roma 29, 81031 Aversa, Italy
[2] Department of Mechanical Engineering, University of Bath, Bath BA2 7AY, UK; ensdasr@bath.ac.uk
* Correspondence: giuliano.destefano@unicampania.it

1. Premise

The present Special Issue consists of a collection of feature articles by distinct investigators and research groups discussing new findings or cutting-edge developments concerning different aspects of fluid mechanics. This Special Issue comprises ten studies covering some of the latest advances in the field, relevant to both theoretical and applied research contexts. The different papers present in-depth analyses, commentaries, and studies of a diverse set of topics within the fluids domain. The main themes include the following:

Reynolds number analysis;
Computational fluid dynamics;
Turbulent flows;
Machine learning and fluid mechanics;
Fluids engineering;
Multidisciplinary optimization in aerodynamics.

Specifically, nine high-quality review papers have been published, along with one original research article. All these works provide detailed insights, opinions, or perspectives on current issues or important trends. In the following section, before presenting the methodologies, the key findings, and the conclusions of each contributing paper, a very brief overview of recent developments and future perspectives in their relevant fields is offered.

2. Feature Articles

2.1. Reynolds Number Analysis

The Reynolds number analysis remains a fundamental tool in fluid mechanics, with ongoing research expanding its applications and enhancing its theoretical foundations. Classically, the Reynolds number serves as a critical parameter in determining flow regimes, particularly in distinguishing between laminar and turbulent flows. Research in this area has advanced significantly, particularly through high-fidelity numerical simulations, e.g., refs. [1,2]. These efforts have also led to the development of improved intermittency and transition models that enhance the accuracy of predicting flow transitions, especially within Reynolds-averaged Navier–Stokes (RANS) frameworks. Additionally, studies on aerodynamic applications have integrated Reynolds number considerations to better predict aircraft performance [3], while emerging standardization efforts aim to unify its application in fields like biomechanics and nanotechnology. Despite recent advances, the influence of this parameter in complex geometry flows (for instance, in the presence of strong curvature or multi-element configurations) is not yet fully understood. Moreover, predicting transition Reynolds numbers under realistic turbulence intensities and environmental conditions still poses challenges, limiting the reliability of current models. There is also

check for updates

Received: 15 May 2025
Accepted: 16 May 2025
Published: 20 May 2025

Citation: De Stefano, G.; Rees, D.A.S. Recent Advances in Fluid Mechanics: Feature Papers, 2024. *Fluids* **2025**, *10*, 137. https://doi.org/10.3390/fluids10050137

Copyright: © 2025 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) license (https://creativecommons.org/licenses/by/4.0/).

a need for better integration of Reynolds number effects across different physical scales and disciplines to foster cross-field consistency [4]. The following two review papers are dedicated to the analysis of this crucial parameter in fluid mechanics.

The work by Saldana et al. [5] explores the historical development and contemporary relevance of the Reynolds number. The paper aims to trace the origins of this parameter, its definition, and its evolution over time, highlighting its significance in various applications across different fields of fluids science and engineering. The study emphasizes the broad applicability of this parameter beyond traditional fluid dynamics, including its relevance in areas such as aerodynamics, microfluidics, and even astrophysics. Recent advancements and reinterpretations of this important number are discussed, including its adaptations for non-Newtonian fluids and granular flows. The authors highlight the importance of the Reynolds number in practical engineering applications, such as the design of pipelines, aircraft, and various fluid mechanics systems, demonstrating its role in optimizing performance and ensuring safety. Finally, they advocate for a deeper understanding of this parameter and its implications in modern fluid dynamics studies.

Similarly, but differently, the work by Tamburrino and Niño [6] examines the universal presence of the Reynolds number and its fundamental role in fluid dynamics, together with its implications across various scientific and engineering fields. This review paper aims to highlight the significance of this parameter as a dimensionless quantity that characterizes flow behavior and its universal applicability in very different contexts. The work explores the application of the Reynolds number for studying a vast range of fluid phenomena, from microscopic scales such as cellular motion to macroscopic scales, such as turbulent aerodynamic and hydrodynamic flows, and even intergalactic dynamics. The recent advancements in the study of this flow parameter are discussed, including its adaptations for non-Newtonian fluids and the exploration of its effects in complex flow scenarios. In conclusion, the Reynolds number represents a fundamental concept in fluid dynamics with widespread implications, with the authors emphasizing the need for continued research to explore its effects in new emerging fields, while reinforcing its relevance in both theoretical and practical contexts.

2.2. Computational Fluid Dynamics

The research findings in computational fluid dynamics (CFD) methodologies have direct implications for various scientific and technological fields. For instance, in aerospace engineering, accurate simulations of turbulent compressible flows are critical for design and analysis. Recent advances in shock-capturing methodologies for high-speed flows apply high-order schemes supplied with improved shock sensors, such as weighted essentially non-oscillatory (WENO) schemes, implicit shock tracking techniques, and adaptive artificial diffusivity, to accurately capture shocks and their interaction with fluid turbulence, e.g., refs. [7,8]. Despite these advances, challenges remain in accurately capturing shock–turbulence interactions at high Reynolds and Mach numbers, especially in three-dimensional, unsteady, and hypersonic regimes. Moreover, computational complexity and numerical dissipation still limit the fidelity of CFD simulations. Future research should focus on developing more efficient and robust high-order schemes capable of handling very complex flows, creating dynamic shock sensors that further reduce numerical dissipation, improving fluid–solid interface modeling under shock impact, and enabling high-fidelity simulations of hypersonic flows. The following two papers review high-order accurate schemes for compressible turbulent flows with applications in geosciences and aerospace engineering.

The work by Assis et al. [9] focuses on the development and evaluation of high-order shock-capturing schemes for simulating high-speed flows. The authors investigate WENO

schemes for simulating the Euler equations under canonical coordinates and the Navier–Stokes equations under generalized coordinates, providing a detailed analysis of their performance, and highlighting their robustness in handling shock waves and discontinuities [10]. Tested for three canonical supersonic flows (upstream of a cylinder, over a blunt wedge, and over a compression ramp), these schemes demonstrated the capability to achieve machine precision, indicating their effectiveness in capturing the complex gas dynamics. Basically, the present results show that these schemes can significantly improve the accuracy and efficiency of simulations compared to traditional low-order methods. One can conclude that high-order accurate shock-capturing schemes are a valuable tool for achieving convergence towards steady states in high-speed flows. However, the importance of continued research to further refine these methods and explore their potential applications in more complex problems should be emphasized.

The work by Yee et al. [11] presents a comprehensive overview of research aimed at improving the predictability and reliability of numerical simulations for compressible flows involving shock waves. The paper focuses on integrating nonlinear dynamical approaches with newly developed high-order numerical methods to quantify numerical uncertainty in simulations of compressible turbulence. In particular, the construction of high-order entropy-conserving, momentum-conserving, and kinetic energy-preserving numerical methods is discussed, while emphasizing the importance of accurate discretization in capturing the complexities of nonlinear systems [12]. As a key finding, the solution space of discrete genuinely nonlinear systems is significantly larger than that of corresponding continuous systems, which can lead to discrepancies in numerical solutions. In fact, traditional uncertainty quantification methods, which often rely on linearized analyses, may not adequately capture the true behavior of fluid flows. The paper highlights the necessity of adaptive numerical methods and the blending of multiple schemes to manage numerical dissipation and enhance stability, particularly in the presence of shock waves [13]. The authors conclude that a nonlinear approach to numerical uncertainty quantification is essential for improving the reliability of CFD calculations. They advocate for continued development of high-order methods and adaptive techniques to better address the challenges posed by complex compressible flows.

The color gradient method (CGM) is a variant of the lattice Boltzmann method (LBM) used for simulating two-phase flow of immiscible fluids through porous media, e.g., ref. [14]. Recent developments have focused on improving the accuracy and stability of the method, where key advances include enhancing isotropy in the color gradient calculation to better handle very high-density ratios, which is critical for realistic two-phase flow modeling [15]. Additionally, numerical improvements in recoloring operators and adaptive grid schemes have been explored to reduce spurious velocities and improve interface capturing. However, despite some progress, gaps remain in fully understanding and optimizing CGM for real-world multiphase flows, especially under extreme conditions such as highly turbulent regimes. Challenges include accurately modeling fluid acceleration effects, viscosity contrasts, and interface dynamics in porous media. Future developments should lead to more robust formulations that maintain accuracy at very high density and viscosity ratios, using improved numerical schemes for interface tracking to minimize spurious currents (and ensure mass conservation), extending the method to complex geometries and porous media with realistic boundary conditions [16], and coupling with advanced turbulence models and machine learning techniques for enhanced simulation fidelity.

In this context, the work by Zahid and Cunningham [17] aims to critically review and evaluate various algorithms and methodologies associated with the CGM approach. The authors compare different methods for modeling fluids with varying viscosities, emphasizing the importance of selecting appropriate algorithms for accurate simulations. The

paper discusses methods for calculating the color gradient, which is essential for modeling the interface between different fluid phases. External forces or accelerations acting on the fluids are also analyzed, with comparisons made between different forcing schemes. The authors highlight the necessity of proper algorithm selection for conducting benchmark simulations, specifically mentioning bubble tests and layered Poiseuille flow as common scenarios for validation. The review indicates that future works will be exploring more complex phenomena such as fluids with widely differing viscosities, wettability effects, and extending the CGM to three-dimensional simulations.

2.3. Turbulent Flows

Accurately predicting the evolution of turbulent flows of practical interest remains a big challenge in CFD, with scientists and engineers constantly developing methods and models to simulate the complex behavior of turbulent flows. In this framework, the integration of classical methodologies such as large eddy simulation (LES) with reduced-order modeling (ROM) appears promising, e.g., refs. [18,19]. Particularly, some recent advances in LES-ROM coupling focus on hybrid models combining physics-based ROM techniques with machine learning (ML) to better capture turbulent flow dynamics. Though LES-inspired closures improve ROM accuracy by modeling unresolved scales, challenges remain in computational cost, generalizability, and the representation of higher modes in proper orthogonal decomposition (POD). Future research should develop efficient nonlinear closures, enhance ML model robustness across flow regimes, incorporate physics constraints to reduce data needs, and extend methods to complex turbulent flows.

The work by Quaini et al. [20] reviews and discusses approaches for effectively bridging these two methodologies to enhance the computational efficiency and accuracy of simulations, particularly for convection-dominated flows. The authors explore various spatial filtering techniques that can be applied to LES to obtain ROM methods, discussing the mathematical foundations of these approaches and their implications for modeling complex fluid behaviors. As a key finding, the integration of LES and ROM can significantly reduce computational costs while maintaining essential flow features, making it suitable for real-time applications and complex geometries. The authors highlight the importance of appropriate filtering techniques in ensuring the stability and accuracy of ROM derived from LES data. They present case studies demonstrating the effectiveness of the proposed methodologies in capturing key flow characteristics in various applications, including biomedical and engineering contexts. The paper also discusses possible future directions for research, emphasizing the need for further development of hybrid models that can seamlessly integrate LES and ROM, also calling for more extensive validation studies to establish the reliability of these approaches in practical scenarios. In conclusion, bridging the two methods through spatial filtering techniques offers a promising pathway for advancing the simulation of complex flows in science and engineering.

For turbulent flows over rough surfaces, recent developments have leveraged advanced simulation techniques such as wall-modeled large-eddy simulations (WM-LES) and direct numerical simulations (DNSs) to investigate complex roughness effects. Some interesting studies have been conducted for flow over heterogeneous roughness, revealing slow turbulence recovery downstream of rough-to-smooth transitions [21,22], while other recent works have highlighted the significant role of porous substrate depth in turbulence penetration and drag characteristics, e.g., ref. [23]. Despite these advances, gaps remain in understanding the detailed mechanisms of turbulence recovery after roughness transitions for heterogeneous roughness patterns, while the interplay between microscale roughness geometry and macroscale permeability in porous substrates is not fully resolved. As these facts limit predictive capabilities for engineering and environmental applications,

predictably, future research will focus on multi-scale modeling that combines different approaches to capture the turbulence behavior in complex flow scenarios.

The review work by Hantsis and Piomelli [24] analyzes how scalar quantities are transported in turbulent flows over rough surfaces. Numerical simulations are utilized to study the velocity and pressure fields in conjunction with scalar transport, exploring different roughness geometries and their effects on the scalar field. The study investigates the existence of a condition analogous to the fully rough regime for scalar transport, similar to what is observed in momentum transfer [25]. The authors highlight the importance of the roughness sublayer, which extends a few roughness heights above the roughness crest, in determining scalar transport characteristics. The paper discusses how Townsend's similarity hypothesis, which applies to momentum transport, can also extend to scalar transport, suggesting similitudes in the mechanisms governing these processes, and implying the differences due to roughness are limited to the roughness sublayer. Novel findings regarding the interactions between turbulent structures and scalar fields in the presence of roughness are presented, contributing to a deeper understanding of transport phenomena in practical engineering applications. The review concludes with a discussion of relevant open questions, emphasizing that, beyond extending the range of simulated Reynolds and Prandtl numbers, the systematic coverage of different roughness types and topologies would be required, as scalar transport appears to remain sensitive to the geometrical details.

2.4. Machine Learning and Fluid Mechanics

Recently, there has been a tremendous increase in the application of machine learning as a new and efficient mathematical tool in fluid mechanics research. In particular, deep reinforcement learning (DRL) has become popular for active flow control, optimization, and automation, leveraging its ability to handle complex, nonlinear, and high-dimensional problems without explicit physical models [26,27]. DRL frameworks use neural networks to learn control policies through interactions with fluid systems, enabling adaptive and efficient flow manipulation. For example, important studies demonstrate the success of this technique in stabilizing vortex shedding, reducing drag, and suppressing vortex-induced vibrations, e.g., ref. [28]. Despite these advances, however, challenges remain in sensor placement and data efficiency, the interpretability of learned policies, and robustness across varying flow regimes. Moreover, training DRL agents is computationally demanding, and transferring strategies from simulations to real-world turbulent flows is still limited. As embedding physical constraints and ROM techniques into DRL frameworks is underexplored but promising for improving generalization and reducing data needs, future research should focus on this aspect to enhance learning efficiency and interpretability.

The work by Kim et al. [29] presents a comprehensive review of the application of DRL in fluid dynamics. The authors categorize the diverse applications of DRL, discussing the methodologies used and the challenges faced in implementing these techniques for fluids. The paper explores recent advancements in applying DRL to fundamental fluids engineering problems, for instance, flow control, including vortex shedding and drag reduction in turbulent flows, demonstrating its capability to adaptively manage complex fluid behaviors. The potential of DRL in shape optimization, where it can be used to enhance aerodynamic performance by optimizing the geometry of airfoils and other structures, is highlighted. Additionally, the automation of CFD processes using DRL is emerging as a promising area, allowing for more efficient simulations and mesh generation, thus reducing computational complexity. In addition, emerging research trends are discussed, such as the use of multi-agent reinforcement and transfer learning to improve the generalizability and efficiency of DRL applications in fluid dynamics. In conclusion, DRL offers significant potential for advancing fluids research by providing innovative solutions to very complex

problems. However, the need for further research addressing challenges related to the DRL implementation and expanding its applicability across various flow scenarios should be emphasized.

2.5. Fluids Engineering

The fluid dynamics of disease transmission is an emergent research area, where rich multi-scale flow physics, from interfacial to multiphase complex flows, play crucial roles [30]. The complex scenario includes the generation and aerosolization of virus-laden respiratory droplets from a host, airborne dispersion and deposition on surfaces, and subsequent inhalation of these bioaerosols by unsuspecting recipients [31]. Fluid mechanics is also key to preventative measures such as the use of face masks, ventilation of indoor environments, and even social distancing [32]. Recent advances with investigative techniques, including new developments in computational and experimental methods, as well as the physical interpretation of results, enhance the knowledge and understanding of fluid flow phenomena in engineered systems and processes in this particular area.

The work by Sankurantripati and Duchaine [33] focuses on fluids engineering technology for improving indoor air quality to mitigate the spread of airborne diseases. Their contribution reviews various ventilation strategies and the role of portable air purifiers using ultraviolet (UV) light in controlling airborne pathogens. Both natural and mechanical systems, and their effectiveness in reducing pathogen concentrations in indoor environments, are analyzed, specifically examining the technologies used in portable air purifiers that incorporate UV-C radiation. As a key finding, effective ventilation systems are demonstrated to significantly lower pathogen concentrations by improving airflow in enclosed spaces, where the integration of portable purifiers with existing ventilation systems is recommended for enhanced control. The study highlights challenges in the practical implementation of UV air purifiers, including the need for further research to both validate their efficacy in real-world applications and optimize their design for different environments. In particular, the importance of understanding the interaction between UV dosage, airflow rates, and the effectiveness of air purifiers is emphasized. The authors conclude that combining effective ventilation strategies with portable purifiers serves as a viable solution for improving indoor air quality and reducing the risk of airborne disease transmission. However, they advocate for continued research to better understand the dynamics of indoor air quality and the role of advanced purification technologies.

2.6. Multidisciplinary Optimization in Aerodynamics

Modern aircraft design technologies increasingly face various complex operating conditions, and one must consider the requirements of different disciplines to achieve good overall performance. Recent advancements in multidisciplinary design optimization (MDO) for aircraft combine high-fidelity CFD analysis with structural, propulsion, and operational models to address complex design challenges, e.g., refs. [34,35]. In particular, the shape design of stealth aircraft needs to give consideration to both aerodynamic and stealth performance, while balancing these two different disciplines [36].

The original research work by Thoulon et al. [37] focuses on the development of a methodology for simultaneously optimizing the aerodynamic performance and stealth characteristics of a fighter aircraft. The paper presents a gradient-based bidisciplinary optimization approach that balances aerodynamic efficiency and low observability for simplified aircraft design. The authors utilize a computational framework that incorporates shape parameterization, through computer-aided design (CAD) modeling, and optimization algorithm, with the sequential least squares quadratic programming method being employed to find optimal designs based on multiple criteria. The optimization process in-

volves both aerodynamic performance, which is evaluated through drag calculations using RANS calculations, and stealth characteristics, where the radar cross-section is analyzed to minimize the visibility of the aircraft to radar systems. The study identifies a Pareto front illustrating the trade-offs between the two different goals, allowing designers to select optimal configurations based on specific mission requirements. The proposed optimization methodology effectively balances diverse requirements, providing valuable insights for the design of next-generation aircraft.

3. Conclusions

This editorial concisely captures the essence of this Special Issue, entitled *Recent Advances in Fluid Mechanics: Feature Papers, 2024*, while offering a brief overview of the relevant research areas of the different contributions. The guest editors consider the present collection to be a precious forum to review and disseminate important research findings and share innovative ideas in the broad field of fluid mechanics. In the same vein, with the next Special Issue of the series, entitled *Feature Reviews for Fluids 2025–2026*, the editors will continue to present both review articles of archival significance and new theoretical, numerical, and experimental studies of lasting scientific value for the fluid dynamics research community.

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

References

1. Nabae, Y.; Inagaki, K.; Kobayashi, H.; Gotoda, H.; Fukagata, K. Large-eddy simulation of high-Reynolds-number turbulent channel flow controlled using streamwise travelling wave-like wall deformation for drag reduction. *J. Fluid Mech.* **2025**, *1003*, A2. [CrossRef]
2. Araya, G.; Lagares-Nieves, C.J.; Jansen, K. Reynolds number dependency in compressible crossflow-jet problems. In Proceedings of the AIAA Science and Technology Forum and Exposition 2025, Orlando, FL, USA, 6–10 January 2025.
3. Smith, L.; Taira, K. The effect of Reynolds number on the separated flow over a low-aspect-ratio wing. *J. Fluid Mech.* **2024**, *992*, R2. [CrossRef]
4. Brynjell-Rahkola, M.; Duguet, Y.; Boeck, T. Route to turbulence in magnetohydrodynamic square duct flow. *Phys. Rev. Fluids* **2025**, *10*, 023903. [CrossRef]
5. Saldana, M.; Gallegos, S.; Gálvez, E.; Castillo, J.; Salinas-Rodríguez, E.; Cerecedo-Sáenz, E.; Hernández-Ávila, J.; Navarra, A.; Toro, N. The Reynolds number: A journey from its origin to modern applications. *Fluids* **2024**, *9*, 299. [CrossRef]
6. Tamburrino, A.; Niño, Y. The universal presence of the Reynolds number. *Fluids* **2025**, *10*, 117. [CrossRef]
7. Li, Q.; Yokoi, K.; Xie, Z.; Omar, S.; Xue, J. A fifth-order high-resolution shock-capturing scheme based on modified weighted essentially non-oscillatory method and boundary variation diminishing framework for compressible flows and compressible two-phase flows. *Phys. Fluids* **2021**, *33*, 056104. [CrossRef]
8. Liang, T.; Fu, L. A new high-order shock-capturing TENO scheme combined with skew-symmetric-splitting method for compressible gas dynamics and turbulence simulation. *Comput. Phys. Commun.* **2024**, *302*, 109236. [CrossRef]
9. Assis, J.C.; Santos, R.D.; Schuabb, M.S.; Falcão, C.E.G.; Freitas, R.B.; Alves, L.S.B. Convergence towards high-speed steady states using high-order accurate shock-capturing schemes. *Fluids* **2024**, *9*, 133. [CrossRef]
10. Wang, Y.; Chen, J.; Wang, Y.; Zeng, Y.; Ke, S. A WENO-based upwind rotated lattice Boltzmann flux solver with lower numerical dissipation for simulating compressible flows with contact discontinuities and strong shock waves. *Appl. Sci.* **2024**, *14*, 11450. [CrossRef]
11. Yee, H.C.; Sweby, P.K.; Sjögreen, B.; Kotov, D.V. A nonlinear approach in the quantification of numerical uncertainty by high-order methods for compressible turbulence with shocks. *Fluids* **2024**, *9*, 250. [CrossRef]
12. Kuja, Y.; Kawai, S. High-order accurate kinetic-energy and entropy preserving (KEEP) schemes on curvilinear grids. *J. Comput. Phys.* **2021**, *442*, 110482.
13. Ge, X.; Vasilyev, O.V.; De Stefano, G.; Hussaini, M.Y. Wavelet-based adaptive unsteady Reynolds-averaged Navier–Stokes simulations of wall-bounded compressible turbulent flows. *AIAA J.* **2020**, *58*, 1529–1549. [CrossRef]
14. Leclaire, S.; Reggio, M.; Trépanier, J.-Y. Isotropic color gradient for simulating very high-density ratios with a two-phase flow lattice Boltzmann model. *Comput. Fluids* **2011**, *48*, 98–112. [CrossRef]

15. Ba, Y.; Liu, H.; Li, Q.; Kang, Q.; Sun, J. Multiple-relaxation-time color-gradient lattice Boltzmann model for simulating two-phase flows with high density ratio. *Phys. Rev. E* **2016**, *94*, 023310.
16. Leclaire, S.; Parmigiani, A.; Malaspinas, O.; Chopard, B.; Latt, J. Generalized three-dimensional lattice Boltzmann color-gradient method for immiscible two-phase pore-scale imbibition and drainage in porous media. *Phys. Rev. E* **2017**, *95*, 033306. [CrossRef] [PubMed]
17. Zahid, F.; Cunningham, J.A. Review of the color gradient lattice Boltzmann method for simulating multi-phase flow in porous media: Viscosity, gradient calculation, and fluid acceleration. *Fluids* **2025**, *10*, 128. [CrossRef]
18. Xie, X.; Mohebujjaman, M.; Rebholz, L.G.; Iliescu, T. Data-driven filtered reduced order modeling of fluid flows. *SIAM J. Sci. Comput.* **2018**, *40*, B834–B857. [CrossRef]
19. Hijazi, S.; Stabile, G.; Mola, A.; Rozza, G. Data-driven POD-Galerkin reduced order model for turbulent flows. *J. Comput. Phys.* **2020**, *416*, 109513. [CrossRef]
20. Quaini, A.; San, O.; Veneziani, A.; Iliescu, T. Bridging large eddy simulation and reduced-order modeling of convection-dominated flows through spatial filtering: Review and perspectives. *Fluids* **2024**, *9*, 178. [CrossRef]
21. Salomone, T.; Piomelli, U.; De Stefano, G. Wall-modeled and hybrid large-eddy simulations of the flow over roughness strips. *Fluids* **2023**, *8*, 10. [CrossRef]
22. Jensen, R.M.; Forooghi, P. A study of turbulent flow over patchy roughness. *J. Fluid Mech.* **2025**, *1006*, A28. [CrossRef]
23. Hao, Z.; García-Mayoral, R. Turbulent flows over porous and rough substrates. *J. Fluid Mech.* **2025**, *1008*, A1. [CrossRef]
24. Hantsis, Z.; Piomelli, U. Numerical simulations of scalar transport on rough surfaces. *Fluids* **2024**, *9*, 159. [CrossRef]
25. Li, Q.; Bou-Zeid, E. Contrasts between momentum and scalar transport over very rough surfaces. *J. Fluid Mech.* **2019**, *880*, 32–58. [CrossRef]
26. Garnier, P.; Viquerat, J.; Rabault, J.; Larcher, A.; Kuhnle, A.; Hachem, E. A review on deep reinforcement learning for fluid mechanics. *Comput. Fluids* **2021**, *225*, 104973. [CrossRef]
27. Viquerat, J.; Meliga, P.; Larcher, A.; Hachem, E. A review on deep reinforcement learning for fluid mechanics: An update. *Phys. Fluids* **2022**, *34*, 113101. [CrossRef]
28. Rabault, J.; Kuchta, M.; Jensen, A.; Réglade, U.; Cerardi, N. Artificial neural networks trained through deep reinforcement learning discover control strategies for active flow control. *J. Fluid Mech.* **2019**, *865*, 281–302. [CrossRef]
29. Kim, I.; Jeon, Y.; Chae, J.; You, D. Deep reinforcement learning for fluid mechanics: Control, optimization, and automation. *Fluids* **2024**, *9*, 216. [CrossRef]
30. Bourouiba, L. The fluid dynamics of disease transmission. *Annu. Rev. Fluid Mech.* **2021**, *53*, 473–508. [CrossRef]
31. Mittal, R.; Ni, R.; Seo, J.-H. The flow physics of COVID-19. *J. Fluid Mech.* **2020**, *894*, F2. [CrossRef]
32. Krishnaprasad, K.A.; Salinas, J.S.; Zgheib, N.; Balachandar, S. Fluid mechanics of air recycling and filtration for indoor airborne transmission. *Phys. Fluids* **2023**, *35*, 013344. [CrossRef]
33. Sankurantripati, S.; Duchaine, F. Indoor air quality control for airborne diseases: A review on portable UV air purifiers. *Fluids* **2024**, *9*, 281. [CrossRef]
34. Wahler, N.F.; Maruyama, D.; Elham, A. Credibility-based multidisciplinary design optimization of electric aircraft. In Proceedings of the AIAA Science and Technology Forum and Exposition 2023, National Harbor, MD, USA, 23–27 January 2023.
35. Ruh, M.L.; Warner, M.; Hwang, J.T. Toward large-scale multidisciplinary design optimization of aircraft with CFD and mixed-fidelity system models. In Proceedings of the AIAA Science and Technology Forum and Exposition 2025, Orlando, FL, USA, 6–10 January 2025.
36. Li, M.; Chen, J.; Zhou, H.; Chen, Y.; Bai, J. Adjoint-based aero/stealth optimization design for UAV. *Acta Aeronaut. Astronaut. Sin.* **2024**, *45*, 530010.
37. Thoulon, C.; Roge, G.; Pironneau, O. Gradient-based aero-stealth optimization of a simplified aircraft. *Fluids* **2024**, *9*, 174. [CrossRef]

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