

Editorial

Industrial CFD and Fluid Modeling in Engineering: Third Edition

Francesco De Vanna ^{1,2} 

¹ Dipartimento di Ingegneria Industriale, Università degli Studi di Padova, Via Venezia 1, 35131 Padova, Italy; francesco.devanna@unipd.it

² Centro di Ateneo di Studi e Attività Spaziali (CISAS) “Giuseppe Colombo”, Università degli Studi di Padova, Via Venezia 15, 35131 Padova, Italy

This third edition of the Special Issue “Industrial CFD and Fluid Modeling in Engineering” continues a line of inquiry begun in two earlier collections, in which computational fluid dynamics (CFD) is examined not as an abstract numerical exercise but as a working instrument of engineering practice. The first edition assembled foundational approaches, with particular attention to the validation of Reynolds-averaged models in realistic geometries; the second edition broadened the scope toward hybrid turbulence modeling, design optimization, multiphase simulation, and the integration of CFD with machine learning and co-simulation. The present edition extends that trajectory with nine new contributions that, taken together, trace the contemporary frontier of industrial CFD. They range from the fidelity of turbulence closures and the mathematical rigor of two-phase formulations, through the vortex-dominated physics that pervades real machinery, flow control, aerodynamic optimization, and the embedding of CFD within digital-twin and reduced-order frameworks [1,2]. The collected studies span an unusually wide range of sectors, namely wind and nuclear energy, molten-salt thermal systems, internal-combustion engines, road and marine transport, and sugar processing. They originate from research groups across Europe, the Americas, and Asia, reflecting the genuinely global character of the field.

Several threads run through the collection. The first is an enduring concern with the fidelity of turbulence modeling and an honest appraisal of what a simulation can and cannot deliver, a concern that links this edition directly to its predecessors. The second is the ubiquity of vortex dynamics: from canonical cylinder wakes to the interior of an engine brake valve and the manifolds of a heat exchanger, organized vortical structures recur as the decisive physics governing performance. The third is the increasingly tight coupling of CFD with optimization algorithms, reduced-order models, and digital twins, through which high-fidelity simulation is translated into fast, deployable engineering tools [3]. In what follows, the nine contributions focused on these issues are discussed.

The edition opens, fittingly, with a study that deepens a line of inquiry begun in the previous collection. In “Boundary Layer Separation from a Curved Backward-Facing Step Using Improved Delayed Detached-Eddy Simulation” (IDDES), McConnell et al. [4], whose earlier contribution to the second edition assessed the same approach for vortex shedding behind a triangular bluff body, turn to the considerably more demanding case in which detachment is governed by an adverse pressure gradient rather than a fixed geometric edge [5–8]. Working in a commercial solver and injecting inlet turbulence through the synthetic eddy method, with profiles drawn from a precursor RANS channel computation [9], they show that IDDES based on the $k-\omega$ SST closure [10] predicts the separation length to within 10.4% of reference large-eddy simulation data, a marked



Received: 25 June 2026

Accepted: 29 June 2026

Published: 10 July 2026

Copyright: © 2026 by the author.

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.

improvement over the 36.2% error returned by RANS, while requiring only about a quarter of the cells demanded by LES. Their careful attention to numerics, demonstrating that a bounded central scheme outperforms the default hybrid scheme, underscores a recurring lesson of industrial CFD: that the choice of discretization can matter as much as the choice of turbulence model [11]. The outcome is an affordable, validated route to separation prediction on curved surfaces of direct relevance to aerofoils, wind- and tidal-turbine blades, and compressor aerodynamics.

Where McConnell et al. [4] demonstrate how far a well-posed hybrid method can be pushed, Yao et al. [12] offer an equally valuable counterpoint: a candid, developer's-eye assessment of the limits of a self-developed industrial solver. In "Numerical Simulation of the Flow Around Cylinders for a Wide Range of Reynolds Numbers", they validate their in-house marine hydrodynamic code on the canonical flow around a circular cylinder across an exceptionally wide Reynolds-number range, from unity to ten million, finding that an unstructured-grid laminar treatment reproduces the mean drag to within 5% at low and subcritical conditions (an error of only 2.7% at $Re = 10^4$), whereas SST $k-\omega$ schemes may underpredict the drag by as much as 45.8% [13–15]. Most instructively, in the transitional band around the drag crisis every scheme tested, including a leading commercial code run under identical conditions, fails to capture the abrupt fall in drag. Through a Taylor-expansion analysis of the near-wall velocity the authors trace this failure to the inability of second-order, finite-volume discretizations to resolve the third-order secondary-flow profiles near separation. By documenting precisely where their solver, and by extension the prevailing class of RANS solvers, breaks down, this work furnishes a constructive roadmap for the development of next-generation industrial codes for marine and offshore engineering.

Complementing these studies of turbulence-model fidelity, Ram et al. [16], in "A Verification of the Two-Fluid Model with Interfacial Inertial Coupling", address rigor of a more fundamental kind: the mathematical well-posedness of the two-fluid model that underpins much of industrial two-phase simulation. When interfacial inertial coupling is represented improperly, the model's momentum equations become elliptic and ill-posed, so that short-wavelength perturbations grow without bound and results are stabilized only by artificial regularization or coarse meshes [17,18]. Adopting a variational formulation cast in terms of superficial velocity and drift flux [19], the authors show that the relative-motion mode reduces, under a fixed-flux approximation, to a Burgers equation, yielding exact analytical solutions that serve as rigorous verification benchmarks. Implementing the inertial-coupling terms as custom source terms in a commercial CFD code, they recover converged material shock waves that match the analytic solution at millimeter resolution without any regularization. In carefully distinguishing verification (solving the equations correctly) from validation, and in showing that physically consistent inertial coupling can be embedded in a general-purpose solver, this contribution strengthens the foundations on which reliable two-phase simulation of nuclear and gas-liquid systems must rest.

While the opening studies concern how faithfully simulations reproduce reality, the next group reminds us how subtle that reality can be. In "Vortex Dynamics Effects on the Development of a Confined Turbulent Wake", Kalogirou et al. [20] contribute a meticulous experimental investigation, rare and welcome in a field increasingly dominated by computation, of how strong wall confinement reshapes the turbulent wake of a circular cylinder. Combining multi-sensor, hot-wire anemometry with laser-sheet visualization across blockage ratios of 7% and 14% in the subcritical regime, they show that the near wake retains a clear Kármán shedding signature at a Strouhal number near 0.2 up to some sixty diameters downstream, beyond which periodicity collapses and the spectra broaden toward channel-like turbulence [21–24]. Confinement is found to invert the relative po-

sitioning of the shed vortices and to narrow the wake, the half-width growth exponent falling from about 0.46 at 7% blockage to 0.24 at 14%, while higher-order moments lag behind the mean velocity in approaching the channel-flow state. The study demonstrates that a confined wake does not relax independently toward a canonical asymptotic state but preserves the imprint of its near-field vortex formation and, in doing so, provides a valuable benchmark dataset against which confined-wake CFD can be validated.

The vortex-dominated physics charted in canonical form by Kalogirou et al. [20] reappears, in far more violent guise, inside the machinery studied by Mo et al. [25] In “Flow Mechanism Analysis of Engine Valve Deviation Under Braking Conditions”, they examine the lateral deviation of an exhaust valve in a heavy-duty diesel engine equipped with a compression-release brake. Under downhill braking a single exhaust valve is held barely open, with a lift of little more than a millimeter, and is observed to sway laterally in a manner that degrades sealing and shortens component life. Through a three-dimensional transient compressible simulation closed with the SST $k-\omega$ model, the authors reveal that the roughly 3.5 MPa pressure difference across the valve drives a near-Mach-3 jet [26,27] and an intricate, asymmetric multi-vortex structure that evolves along the valve axis through three distinct zones and produces a net lateral force responsible for the observed deviation. By localizing the force generation to the valve’s curved section and cylindrical body, the study supplies the physical understanding required to inform structural and flow-control remedies. It is an instructive example of CFD deployed not to optimize a design but to diagnose the root cause of a stubborn reliability problem.

That vortices are not merely a physical curiosity but a quantifiable source of engineering loss is made explicit by Wang et al. [28] In “A Vortex-Induced Correction Method for Pressure Loss Prediction in Fluid Network Theory”, they confront a systematic shortcoming of one-dimensional fluid-network theory: its tendency to underestimate pressure losses, frequently by more than 20%, in the parallel pipe bundles and manifolds of molten-salt heat exchangers, where three-dimensional vortices dissipate energy that lumped resistance models cannot represent [29–31]. Coupling three-dimensional CFD, validated against published flow-distribution experiments to within 10%, with conventional network analysis, they introduce a vortex-induced additional loss coefficient obtained by least-squares fitting of the simulation data and embedded directly into the Darcy–Weisbach friction term. A systematic parametric study shows that low viscosity, high inlet velocity, a large manifold cross-section, and counter-flow arrangements all intensify the vortices and the prediction error, with uncorrected deviations reaching 23–31% at low viscosity; the fitted coefficient, ranging from 1.15 to 1.37, reduces the error to within 5%. The approach preserves the speed of network theory while restoring much of the accuracy of full CFD, exemplifying the reduced-order strategies on which the design of thermal-energy systems increasingly depends.

The final group turns from physical understanding to the translation of CFD into tangible design and operational value. In “Passive Control of Boundary-Layer Separation on a Wind Turbine Blade Using Varying-Parameter Flow Deflector”, Chen et al. [32] address the perennial problem of separation near the root of horizontal-axis wind-turbine blades, where three-dimensional effects and high local angles of attack erode power and aggravate fatigue. Recognizing that the local inflow varies along the span, they devise a varying-parameter passive flow deflector whose normal position and installation angle change spanwise so as to match the local angle of attack at each section, yielding an adaptable, energy-free device in place of a fixed one [33–35]. The aerodynamics are resolved with three-dimensional RANS computations of the NREL Phase VI rotor using the transition SST $k-\omega$ model [36,37], and the deflector geometry is optimized through a Box–Behnken response-surface model whose quadratic fit attains an R^2 of 0.98 [3,38]. The optimized

deflector raises blade shaft torque by 24.7% at 10 m s^{-1} , with the controlled-blade CFD result matching the surrogate prediction to within 0.1%, and delivers speed-dependent gains across the operating range. The work shows how response-surface optimization and passive flow control can be combined to recover a substantial fraction of the aerodynamic performance lost to root separation.

A complementary translation of CFD into operational insight is offered by Alic et al. [39] In “Transient CFD Study of Aerodynamic Interaction Between Heavy-Duty Trucks During Highway Merging and Platoon Formation Under Crosswind”, they study the aerodynamic interaction between two heavy-duty trucks as a trailing vehicle merges from an acceleration lane and forms a platoon. Departing from the steady-state analyses that dominate platooning research, they solve the unsteady RANS equations with a dynamic deforming mesh so that wake development and instantaneous loads are resolved as the inter-vehicle gap evolves [40–43]. The simulations show that in still air the trailing truck attains about a 40% reduction in drag, but that a crosswind producing roughly a 20° yaw angle nearly halves this benefit to about 21%, while raising the leading truck’s drag by some 15% and reducing the trailing truck’s side force by about 31% through partial windbreak shielding. By identifying three distinct interaction regimes and a non-linear, saturating relationship between spacing and drag, the study makes clear that unsteady and crosswind effects are indispensable to a realistic assessment of platoon performance and to the design of robust cooperative-control strategies.

The edition closes with a study that embeds CFD within the operational fabric of an industrial plant. In “Digital Twins: A Solution Under the Standard k – ϵ Model in Industrial CFD, to Predict Ideal Conditions in a Sugar Dryer”, Guerrero-Hernández et al. [44] construct a digital twin of a centrifugal sugar dryer, an industrial process that conventionally runs without control and relies on operator experience alone, by coupling a thermodynamic energy balance with a three-dimensional CFD model of the turbulent, heated airflow. Solving the Navier–Stokes and energy equations with the standard k – ϵ model, and validating the predictions against live distributed-control-system data to within 5%, they show that an optimized configuration stabilizes the dryer at a uniform air temperature near 85°C , holds the sugar between 30 and 35°C for safe packaging, and lowers the steam internal energy enough to save 4.25% of steam energy. That the venerable k – ϵ closure proves entirely adequate here is itself a useful reminder that model sophistication should be matched to the problem at hand. By diagnosing out-of-range variables and prescribing efficient set-points in real time without halting production, the study illustrates how a CFD-driven digital twin can carry the discipline from analysis into the continuous operation of a manufacturing process.

Viewed against the prevailing state of the art, the collection is perhaps best read as a concerted response to a set of standing knowledge gaps in the field, each contribution closing a specific deficiency in the existing literature. The first gap concerns the fidelity of turbulence closures and a candid reckoning with the limits of present solvers: hybrid RANS–LES had not been verified for adverse-pressure-gradient separation from a curved surface of large radius, a shortfall that McConnell et al. [4] address, while Yao et al. [12] supply both a benchmarked, self-developed marine solver and a developer’s diagnosis of why second-order schemes forfeit the drag crisis. A second gap is mathematical rather than numerical, namely the want of an industrial two-fluid model rendered well-posed without artificial regularization, together with exact solutions against which to verify it, which Ram et al. [16] now provide. A third surrounds vortex dynamics as governing physics, where the link between near-field shedding and far-field momentum transport under strong confinement (Kalogirou et al. [20]), the flow origin of exhaust-valve deviation under braking (Mo et al. [25]), and the unrepresented dissipation of three-dimensional vortices

within one-dimensional network theory (Wang et al. [28]) had each resisted resolution. A fourth and final gap concerns the translation of high-fidelity computation into deployable engineering tools, met here by span-adaptive passive flow control for three-dimensional blades (Chen et al. [32]), transient rather than steady-state platoon aerodynamics under crosswind (Alic et al. [39]), and a CFD-coupled digital twin for real-time plant operation (Guerrero-Hernández et al. [44]). In each instance the present edition does not merely add another simulation but settles a question the field had left open, which is the precise sense in which this Special Issue advances industrial CFD.

Taken together, the nine studies of this third edition of “Industrial CFD and Fluid Modeling in Engineering” chart a coherent arc from the foundations of simulation to its industrial deployment. They sharpen our understanding of turbulence-model fidelity and the limits of present solvers, place the mathematical well-posedness of two-phase formulations on firmer ground, illuminate the vortex dynamics that govern flows from canonical wakes to engine valves and heat-exchanger manifolds, and demonstrate how flow control, surrogate-based optimization, and digital twins convert high-fidelity computation into measurable engineering value. Recurring throughout is a maturing disposition within the field: a willingness to validate honestly, to verify rigorously, and to report not only where CFD succeeds but where it still falls short [1,2]. By advancing both the accuracy and the reach of fluid-flow simulation across energy, transport, and process engineering, these contributions collectively push the boundaries of what industrial CFD can achieve while pointing toward the innovations still to come. As Guest Editor, I extend my sincere gratitude to all the authors for their high-quality contributions, to the reviewers for their rigorous and constructive feedback, and to the editorial team for their invaluable support in bringing this third edition to fruition. Together these efforts have produced a rich and insightful collection that will, I trust, stimulate further research and development in industrial CFD and fluid modeling.

Acknowledgments: As Guest Editor, the author thanks all contributing authors, the reviewers, and the editorial staff of *Fluids* for their dedication in realizing this third edition of the Special Issue.

Conflicts of Interest: The author declares no conflicts of interest.

References

1. Spalart, P.R. Strategies for Turbulence Modelling and Simulations. *Int. J. Heat Fluid Flow* **2000**, *21*, 252–263. [\[CrossRef\]](#)
2. Duraisamy, K.; Iaccarino, G.; Xiao, H. Turbulence Modeling in the Age of Data. *Annu. Rev. Fluid Mech.* **2019**, *51*, 357–377. [\[CrossRef\]](#)
3. Karbasian, H.R.; Vermeire, B.C. Application of Physics-Constrained Data-Driven Reduced-Order Models to Shape Optimization. *J. Fluid Mech.* **2022**, *934*, A32. [\[CrossRef\]](#)
4. McConnell, M.R.; Knight, J.; Buick, J.M. Boundary Layer Separation from a Curved Backward-Facing Step Using Improved Delayed Detached-Eddy Simulation. *Fluids* **2025**, *10*, 145. [\[CrossRef\]](#)
5. Spalart, P.R. Detached-Eddy Simulation. *Annu. Rev. Fluid Mech.* **2009**, *41*, 181–202. [\[CrossRef\]](#)
6. Shur, M.L.; Spalart, P.R.; Strelets, M.K.; Travin, A.K. A Hybrid RANS-LES Approach with Delayed-DES and Wall-Modelled LES Capabilities. *Int. J. Heat Fluid Flow* **2008**, *29*, 1638–1649. [\[CrossRef\]](#)
7. Gritskevich, M.S.; Garbaruk, A.V.; Schütze, J.; Menter, F.R. Development of DDES and IDDES Formulations for the $k-\omega$ Shear Stress Transport Model. *Flow Turbul. Combust.* **2012**, *88*, 431–449. [\[CrossRef\]](#)
8. Travin, A.; Shur, M.; Strelets, M.; Spalart, P. Detached-Eddy Simulations Past a Circular Cylinder. *Flow Turbul. Combust.* **2000**, *63*, 293–313. [\[CrossRef\]](#)
9. Jarrin, N.; Benhamadouche, S.; Laurence, D.; Prosser, R. A Synthetic-Eddy-Method for Generating Inflow Conditions for Large-Eddy Simulations. *Int. J. Heat Fluid Flow* **2006**, *27*, 585–593. [\[CrossRef\]](#)
10. Menter, F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA J.* **1994**, *32*, 1598–1605. [\[CrossRef\]](#) [\[PubMed\]](#)
11. De Vanna, F.; Baldan, G.; Picano, F.; Benini, E. On the Coupling between Wall-Modeled LES and Immersed Boundary Method towards Applicative Compressible Flow Simulations. *Comput. Fluids* **2023**, *266*, 106058. [\[CrossRef\]](#)

12. Yao, H.; Hu, T.; Yang, J.; Wang, J.; Wu, C. Numerical Simulation of the Flow Around Cylinders for a Wide Range of Reynolds Numbers. *Fluids* **2026**, *11*, 68. [[CrossRef](#)]
13. Roshko, A. Experiments on the Flow Past a Circular Cylinder at Very High Reynolds Number. *J. Fluid Mech.* **1961**, *10*, 345–356. [[CrossRef](#)]
14. Lehmkuhl, O.; Rodríguez, I.; Borrell, R.; Chiva, J.; Oliva, A. Unsteady Forces on a Circular Cylinder at Critical Reynolds Numbers. *Phys. Fluids* **2014**, *26*, 125110. [[CrossRef](#)]
15. Rodríguez, I.; Lehmkuhl, O.; Chiva, J.; Borrell, R.; Oliva, A. On the Flow Past a Circular Cylinder from Critical to Super-Critical Reynolds Numbers: Wake Topology and Vortex Shedding. *Int. J. Heat Fluid Flow* **2015**, *55*, 91–103. [[CrossRef](#)]
16. Ram, R.; López-de-Bertodano, M.; Howard, J.A.; Clausse, A. A Verification of the Two-Fluid Model with Interfacial Inertial Coupling. *Fluids* **2025**, *10*, 268. [[CrossRef](#)]
17. Barnea, D.; Taitel, Y. Kelvin-Helmholtz Stability Criteria for Stratified Flow: Viscous Versus Non-Viscous (Inviscid) Approaches. *Int. J. Multiph. Flow* **1993**, *19*, 639–649. [[CrossRef](#)]
18. Fox, R.O.; Laurent, F.; Vié, A. A Hyperbolic Two-Fluid Model for Compressible Flows with Arbitrary Material-Density Ratios. *J. Fluid Mech.* **2020**, *903*, A5. [[CrossRef](#)]
19. Drew, D.A.; Cheng, L.; Lahey, R.T. The Analysis of Virtual Mass Effects in Two-Phase Flow. *Int. J. Multiph. Flow* **1979**, *5*, 233–242. [[CrossRef](#)]
20. Kalogirou, I.D.; Romeos, A.; Giannadakis, A.; Mihalakakou, G.; Panidis, T. Vortex Dynamics Effects on the Development of a Confined Turbulent Wake. *Fluids* **2025**, *10*, 283. [[CrossRef](#)]
21. Williamson, C.H.K. Vortex Dynamics in the Cylinder Wake. *Annu. Rev. Fluid Mech.* **1996**, *28*, 477–539. [[CrossRef](#)]
22. Sahin, M.; Owens, R.G. A Numerical Investigation of Wall Effects up to High Blockage Ratios on Two-Dimensional Flow Past a Confined Circular Cylinder. *Phys. Fluids* **2004**, *16*, 1305–1320. [[CrossRef](#)]
23. Chen, J.H.; Pritchard, W.G.; Tavener, S.J. Bifurcation for Flow Past a Cylinder between Parallel Planes. *J. Fluid Mech.* **1995**, *284*, 23–41. [[CrossRef](#)]
24. Nguyen, Q.D.; Lu, W.; Chan, L.; Ooi, A.; Lei, C. A State-of-the-Art Review of Flows Past Confined Circular Cylinders. *Phys. Fluids* **2023**, *35*, 071301. [[CrossRef](#)]
25. Mo, W.; Dou, Z.; Sun, Q.; Chen, Z. Flow Mechanism Analysis of Engine Valve Deviation Under Braking Conditions. *Fluids* **2026**, *11*, 87. [[CrossRef](#)]
26. Papamoschou, D.; Roshko, A. The Compressible Turbulent Shear Layer: An Experimental Study. *J. Fluid Mech.* **1988**, *197*, 453–477. [[CrossRef](#)]
27. Tam, C.K.W. Supersonic Jet Noise. *Annu. Rev. Fluid Mech.* **1995**, *27*, 17–43. [[CrossRef](#)]
28. Wang, X.; Liang, L.; Song, Q.; Ji, Y.; Sun, M.; Li, H. A Vortex-Induced Correction Method for Pressure Loss Prediction in Fluid Network Theory. *Fluids* **2026**, *11*, 52. [[CrossRef](#)]
29. Bajura, R.A.; Jones, E.H., Jr. Flow Distribution Manifolds. *J. Fluids Eng.* **1976**, *98*, 654–665. [[CrossRef](#)]
30. Hassan, J.M.; Mohamed, T.A.; Mohammed, W.S.; Alawee, W.H. Modeling the Uniformity of Manifold with Various Configurations. *J. Fluids* **2014**, *2014*, 325259. [[CrossRef](#)]
31. Patiño-Jaramillo, G.A.; Iglesias, I.; Vera, M. Laminar Flow and Pressure Loss in Planar Tee Joints: Pressure Loss Coefficients. *Eur. J. Mech. -B/Fluids* **2022**, *94*, 263–275. [[CrossRef](#)]
32. Chen, X.; Qiu, J.; Zhong, J.; Zhang, C.; Gan, Y. Passive Control of Boundary-Layer Separation on a Wind Turbine Blade Using Varying-Parameter Flow Deflector. *Fluids* **2025**, *10*, 270. [[CrossRef](#)]
33. Lin, J.C. Review of Research on Low-Profile Vortex Generators to Control Boundary-Layer Separation. *Prog. Aerosp. Sci.* **2002**, *38*, 389–420. [[CrossRef](#)]
34. Gad-el-Hak, M.; Bushnell, D.M. Separation Control: Review. *J. Fluids Eng.* **1991**, *113*, 5–30. [[CrossRef](#)]
35. Gopalakrishnan Meena, M.; Taira, K.; Asai, K. Airfoil-Wake Modification with Gurney Flap at Low Reynolds Number. *AIAA J.* **2018**, *56*, 1348–1359. [[CrossRef](#)]
36. Fava, T.C.L.; Henningson, D.S.; Hanifi, A. Boundary Layer Stability on a Rotating Wind Turbine Blade Section. *Phys. Fluids* **2024**, *36*, 094128. [[CrossRef](#)]
37. Lee, H.M.; Wu, Y. An Experimental Study of Stall Delay on the Blade of a Horizontal-Axis Wind Turbine Using Tomographic Particle Image Velocimetry. *J. Wind Eng. Ind. Aerodyn.* **2013**, *123*, 56–68. [[CrossRef](#)]
38. Yondo, R.; Andrés, E.; Valero, E. A Review on Design of Experiments and Surrogate Models in Aircraft Real-Time and Many-Query Aerodynamic Analyses. *Prog. Aerosp. Sci.* **2018**, *96*, 23–61. [[CrossRef](#)]
39. Alic, D.D.; Miklos, I.Z.; Miklos, C.C. Transient CFD Study of Aerodynamic Interaction Between Heavy-Duty Trucks During Highway Merging and Platoon Formation Under Crosswind. *Fluids* **2026**, *11*, 119. [[CrossRef](#)]
40. Grandemange, M.; Gohlke, M.; Cadot, O. Turbulent Wake Past a Three-Dimensional Blunt Body. Part 1. Global Modes and Bi-Stability. *J. Fluid Mech.* **2013**, *722*, 51–84. [[CrossRef](#)]

41. Ebrahim, H.; Dominy, R. Wake and Surface Pressure Analysis of Vehicles in Platoon. *J. Wind Eng. Ind. Aerodyn.* **2020**, *201*, 104144. [[CrossRef](#)]
42. Törnell, J.; Sebben, S.; Elofsson, P. Effects of Separation Distance, Lateral Offset, and Yaw on the Drag of a Truck and SUV Platoon. *J. Wind Eng. Ind. Aerodyn.* **2023**, *241*, 105517. [[CrossRef](#)]
43. Tunay, T.; Firat, E.; Sahin, B. Experimental Investigation of the Flow Around a Simplified Ground Vehicle Under Effects of the Steady Crosswind. *Int. J. Heat Fluid Flow* **2018**, *71*, 137–152. [[CrossRef](#)]
44. Guerrero-Hernández, V.; Reyes-Morales, G.; Bastián Lima, V.A.; Ortega-Moody, J.; Bertel, Q.R.Q.; Rodríguez, G.A.; Sánchez, B.E.G.; Ceballos-Díaz, C.; Herazo, L.C.S. Digital Twins: A Solution Under the Standard $k-\epsilon$ Model in Industrial CFD, to Predict Ideal Conditions in a Sugar Dryer. *Fluids* **2025**, *10*, 146. [[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.