Spectrum of Mechanical Engineering and Operational Research, Volume 2, Issue 1 (2025) 13-23

SCIENTIFIC OASIS

Spectrum of Mechanical Engineering
and Operational Research

Journal homepage: www.smeor-journal.org
eISSN: 3042-0288

Numerical Investigation of Large Vehicle Aerodynamics Under the Influence of Crosswind

Daniela Alic[1,*], Aleksandar Miltenovic[2], Milan Banic[2], Raul Vicente Zafra[3]

[1] Department of Engineering and Management, Faculty of Engineering Hunedoara, Politehnica University Timisoara, Romania
[2] Department of Mechanical Engineering, Faculty of Mechanical Engineering, University of Nis, Serbia
[3] Institute of Computer Science, University of Tartu, Estonia

ARTICLE INFO

Article history:
Received 25 June 2024
Received in revised form 9 October 2024
Accepted 2 November 2024
Available online 10 November 2024

Keywords:
Aerodynamic analysis; Crosswind; CFD.

ABSTRACT

This paper presents a numerical analysis aimed at estimating the impact of crosswinds on the aerodynamics of large vehicles using ANSYS Fluid Flow and Static Structural, offering insights into the qualitative behaviour of trucks under typical traffic conditions. The cases analysed focus on airflow distribution when a truck, traveling at approximately 100 km/h, passes under a portal while exposed to varying crosswind intensities: weak, moderate, and strong lateral winds. A detailed examination of the truck and portal reveals significant interactions between the induced airflow patterns, affecting truck handling and stability. This is evidenced by shifts in velocity and pressure distributions, changes in drag coefficients, and variations in turbulent kinetic energy, all of which are analysed and discussed in this study.

1. Introduction

The aerodynamics of heavy-duty vehicle, particularly trucks, is significantly impacted by crosswinds, which can compromise vehicle stability, fuel efficiency, and road safety [1]. Crosswinds generate lateral forces and yawing moments that increase aerodynamic drag and impact vehicle handling, particularly at high speeds or on open roads [2]. Given the critical role of commercial trucking in logistics and transportation systems worldwide, addressing these aerodynamic challenges is essential for heavy-duty trucks [3], where crosswinds also affect nearby infrastructure, including traffic signs and highway structures [4, 5]. As a result, research into vehicle aerodynamics has grown increasingly important, with advanced Computational Fluid Dynamics (CFD) simulations allowing detailed investigations into complex crosswind interactions, flow distribution, drag forces, and induced environmental effects [6].

Numerous studies in vehicle aerodynamics have addressed the impact of crosswinds on vehicle stability and aerodynamic performance [7, 8]. Early research focused on steady-state wind conditions

---

[*] *Corresponding author.*
E-mail address: daniela.alic@upt.ro

https://doi.org/10.31181/smeor21202526

[9], which offered only a limited understanding of crosswind effects and did not fully capture the transient and unsteady nature of crosswinds. Advances in CFD and experimental facilities such as wind tunnels have since facilitated a shift toward investigating transient vehicle dynamics [10, 11]. CFD, particularly through tools like ANSYS software, has emerged as an essential tool for analyzing large-vehicle behavior under crosswind conditions. With robust solvers and advanced meshing capabilities, ANSYS enables detailed studies of flow separation, vortex formation, and pressure distribution around vehicle surfaces—characteristics challenging to observe through physical experiments alone. Recent research in crosswind aerodynamics has focused on optimizing vehicle shapes to reduce aerodynamic drag and enhance stability under crosswind conditions [12, 13]. Streamlining techniques, such as contour adjustments and the addition of aerodynamic devices like deflectors, side skirts, and vortex generators, have been explored to mitigate destabilizing effects of crosswinds. Studies indicate that these devices, when implemented effectively, significantly improve both aerodynamic performance and lateral stability. Field-testing methodologies and wind tunnel setups, incorporating features like moving ground planes and rotating wheels, complement CFD analyses, providing valuable experimental data to validate and refine models.

Another emerging area in crosswind research is the study of Truck-Induced Wind Gusts (TIWG), which occur as large vehicles displace air, generating gusts that can impact nearby objects and road users. TIWG is especially relevant when trucks pass close to roadside structures, such as highway signs and traffic signals, as these gusts may cause structural vibrations, fatigue, and even damage over time [14, 15]. Understanding the aerodynamic impact of these gusts, particularly how they vary with crosswind conditions, is crucial for infrastructure resilience and safety but remains a topic requiring more focused investigation.

Despite significant advancements, there remain critical gaps in understanding the combined effects of crosswinds and truck-induced gusts on both vehicle performance and nearby infrastructure. While previous studies have analyzed crosswind impacts on vehicle stability and drag reduction, fewer studies have simultaneously examined the effects of crosswinds on flow distribution and TIWG on surrounding highway structures. These interactions are complex and require a multi-faceted approach that considers both the vehicle's aerodynamic performance under crosswind conditions and the environmental impact of gusts on roadside infrastructure.

Addressing this gap is essential for developing safer, more efficient trucking solutions and ensuring that highway structures are designed to withstand these transient aerodynamic forces. By focusing on these less explored areas, this research aims to contribute to a more comprehensive understanding of the impact of crosswinds on heavy vehicles and the safety implications for highway infrastructure.

The primary objectives of this research are threefold:

- Flow Distribution Analysis: To examine the impact of crosswinds on the flow distribution around the truck body, identifying regions of high pressure, flow separation, and vortex formation under different crosswind intensities and angles.
- Drag Variation Analysis: To quantify drag force variations experienced by the truck under changing crosswind conditions, providing insights into how these fluctuations influence energy consumption and vehicle handling.
- Estimation of Truck-Induced Wind Gusts (TIWG): To evaluate the effect of TIWG on roadside infrastructure, specifically analyzing the potential impact of truck-induced gusts on traffic signs and highway signal structures. This analysis aims to understand how gust forces generated by the truck interact with crosswinds to create pressures and vibrations on nearby objects, providing insights for infrastructure design.

To address these objectives, a numerical investigation is performed using ANSYS CFD software to analyze the aerodynamic behavior of a truck under crosswind conditions and estimate the influence of the TIWG on a portal-type highway sign support structure.

2. Theoretical basis and Methodology

This section discusses the setup behind the numerical analysis, as well as the steps involved in the structured approach used to solve the mathematical model and capture the physical phenomena behind the aerodynamics of a large vehicle under the influence of crosswinds, in realistic traffic environment.

Numerical modeling with specialized Computational Fluid Dynamics (CFD) software, such as ANSYS Fluid Flow (Fluent with Fluent Meshing) and Static Structural, is widely used in a range of applications within the automotive and mechanical engineering design domains due to its advanced physics modeling capabilities and industry-leading accuracy. Fluent is one of two CFD packages included in the ANSYS mechanical software suite, providing robust tools for addressing complex fluid dynamics challenges. ANSYS Static Structural software is a tool used to analyze the structural response under aerodynamic loads in a steady-state environment. In vehicle aerodynamic analysis, it is often coupled with a Computational Fluid Dynamics (CFD) tool to handle Fluid-Solid Interaction (FSI).

The current analysis employs one-way FSI, which involves simulating the aerodynamic forces on the portal surface without accounting for any feedback effect of the structural response on the fluid flow. The process begins with a Computational Fluid Dynamics (CFD) simulation, where the truck and the portal are placed in a virtual wind tunnel. Here, the aerodynamic loads (pressure and shear forces) on the solid surfaces are calculated by solving the Navier-Stokes equations for the surrounding airflow. The aerodynamic forces calculated from the CFD analysis are then transferred to the structural model of the portal. This transfer is done via coupling interfaces within ANSYS, where only the fluid side sends force data to the structural model without receiving any deformation feedback. With the aerodynamic loads applied to the structural model, a Finite Element Analysis (FEA) simulation is run. Here, the portal's structural response, such as stresses and deformations, is computed based on these aerodynamic forces. Since this is a one-way FSI, the structural deformation does not impact the fluid flow, meaning that the airflow remains unchanged even if the structure deforms. The results from the structural analysis are used to assess areas of high stress or potential deformation, which is crucial for understanding load-bearing capabilities and for optimizing portal design without needing to update the airflow simulation. One-way FSI is computationally less expensive than two-way FSI, making it suitable for initial aerodynamic studies where the structural deformation is minimal and doesn’t significantly alter the fluid flow.

The governing equations for modeling airflow in CFD are the Navier-Stokes equations, which describe the motion of fluids such as air [16]. They are discretized using the Finite Volume Method and evaluated with a Least Squares Cell-Based gradient approach [17]. Our numerical model uses the viscous Shear Stress Transport (SST) k-ω model, a commonly applied two-equation eddy-viscosity turbulence model that approximates the Reynolds-Averaged Navier–Stokes (RANS) equations and incorporates a modified turbulent viscosity formulation to capture the transport effects of primary turbulent shear stress [18]. We employ a Coupled Scheme for Pressure-Velocity Coupling, utilizing a Rhie-Chow momentum-based flux approach [19]. This method offers a robust and efficient single-phase solution for steady-state flows, delivering better performance compared to segregated schemes. To enhance simulation accuracy and effectively capture turbulent flow around the vehicles, second-order upwind schemes are applied to the Turbulent Kinetic Energy and Specific Dissipation

Rate. During the Hybrid initialization phase, the External-Aero Favorable Settings option is enabled to generate a more precise initial velocity field for external-aero simulations. The volume mesh is composed of poly-hexcore elements, with localized refinement around the vehicles, following the standard watertight geometry workflow in ANSYS Fluent Meshing.

The Static Structural analysis relies on fundamental equations from continuum mechanics to analyze the behavior of structures under static (non-time-dependent) loads. The equilibrium equations, also known as force balance equations, ensure that the sum of forces and moments in the system is zero. The strain-displacement equations relate the displacements in the material to the strains, which represent the deformation in the structure. The constitutive equations describe how the material responds to stress, based on Hooke's law. As boundary conditions we specify constraints in the form of fixed supports at the base of the portal, and imported loads applied to the whole structure, as distributed pressure, as resulted from the Fluent calculation. These conditions are essential for solving our problem, as they define how the structure interacts with its environment. ANSYS Static Structural uses the Finite Element Method (FEM) to discretize the structure into smaller, manageable elements. The above equations are applied to each element, and the displacements, strains, and stresses are computed for the entire structure. The solution method involves assembling a global stiffness matrix and solving a system of linear equations to find the displacement field, which then allows for the calculation of strain and stress.

3. Numerical analysis

3.1 Numerical modelling

To simultaneously examine the aerodynamic behavior of the truck and highlight the influence of Truck-Induced Wind Gusts on the overhead span sign structure under crosswind conditions, the numerical investigation was conducted in two traffic phases. In the first phase, the truck travels in the open area in the first lane, approaching the overhead span sign structure (Fig. 1a). In the second phase, the truck passes beneath the overhead span sign structure, reaching the point where the front of the truck aligns with the right column of the portal frame (Fig. 1b).

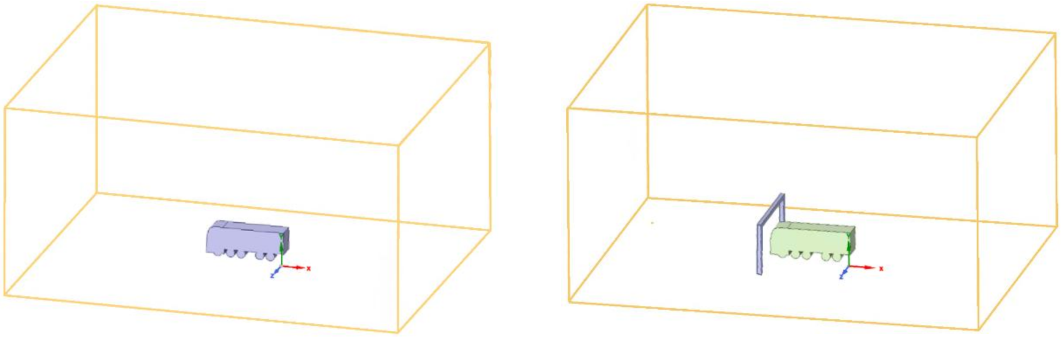

(a) (b)

Fig. 1. Analyzed Traffic Phases: (a) truck traveling in an open area on the first lane (b) truck passing beneath the overhead span sign structure

The numerical simulation domain is a box measuring 50 m in length, 40 m in width, and 24 m in height. The boundary conditions are configured as follows: the front yz boundary allows inflow in the +x direction as a velocity inlet, while the right yx boundary allows inflow in the +z direction. Pressure outlets are set at the back yz and left yx boundaries. The top xz boundary is modeled as a stationary, zero-shear wall, while the ground xz boundary is a moving no-slip wall. Both the portal and truck surfaces are stationary no-slip walls. In viscous fluids, the flow velocity at solid surfaces is zero relative to the boundary, satisfying the no-slip condition. This condition arises because the adhesive force between the fluid and solid particles is stronger than the cohesive forces within the fluid itself.

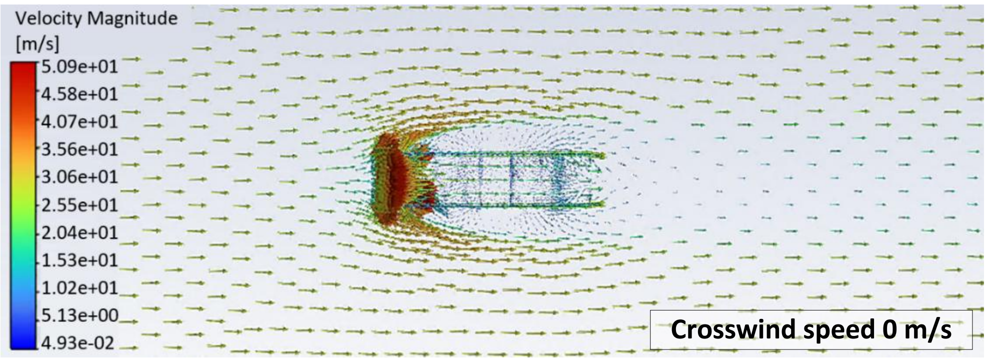
The truck with the streamlined geometry of a European truck [20] has a length of 10.5 m, width of 2.5 m and height of 4 m, is traveling at a speed of 28 m/s (approximately 100 km/h), modeled by an airflow entering through the front inlet boundary and moving in the positive x-direction (Fig. 1). The portal stands 6.5 m high with a span of 12.5 m and is constructed from Structural Steel beams, each with a thickness of 0.5 m. The truck travels in the first lane, positioned 2.5 m away from the portal's pillar.

The velocity of the crosswind, entering through the right inlet boundary and directed along the positive z-axis, varies among 5 m/s, 10 m/s, and 15 m/s, representing weak, moderate, and strong wind conditions, respectively. The fluid is air, with a density of 1.225 kg/m³ and a viscosity of 1.7894×10^{-5} kg/(m·s). The turbulent intensity, defined as the percentage variation in flow rate relative to the average flow rate, is set to 1%, while the turbulent viscosity ratio at the boundaries is set to 2.

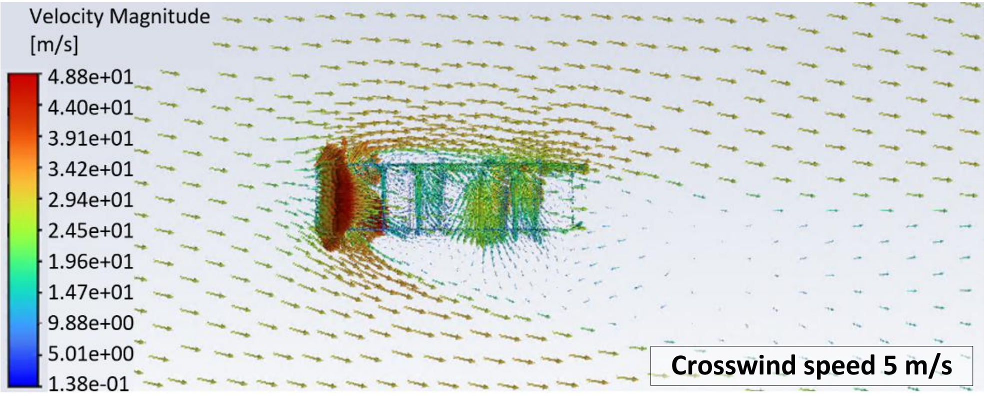
3.2 Numerical results, interpretations and discussions

The numerical results of the aerodynamic behavior analysis under crosswind conditions are presented for the two traffic phases outlined in Fig. 1.

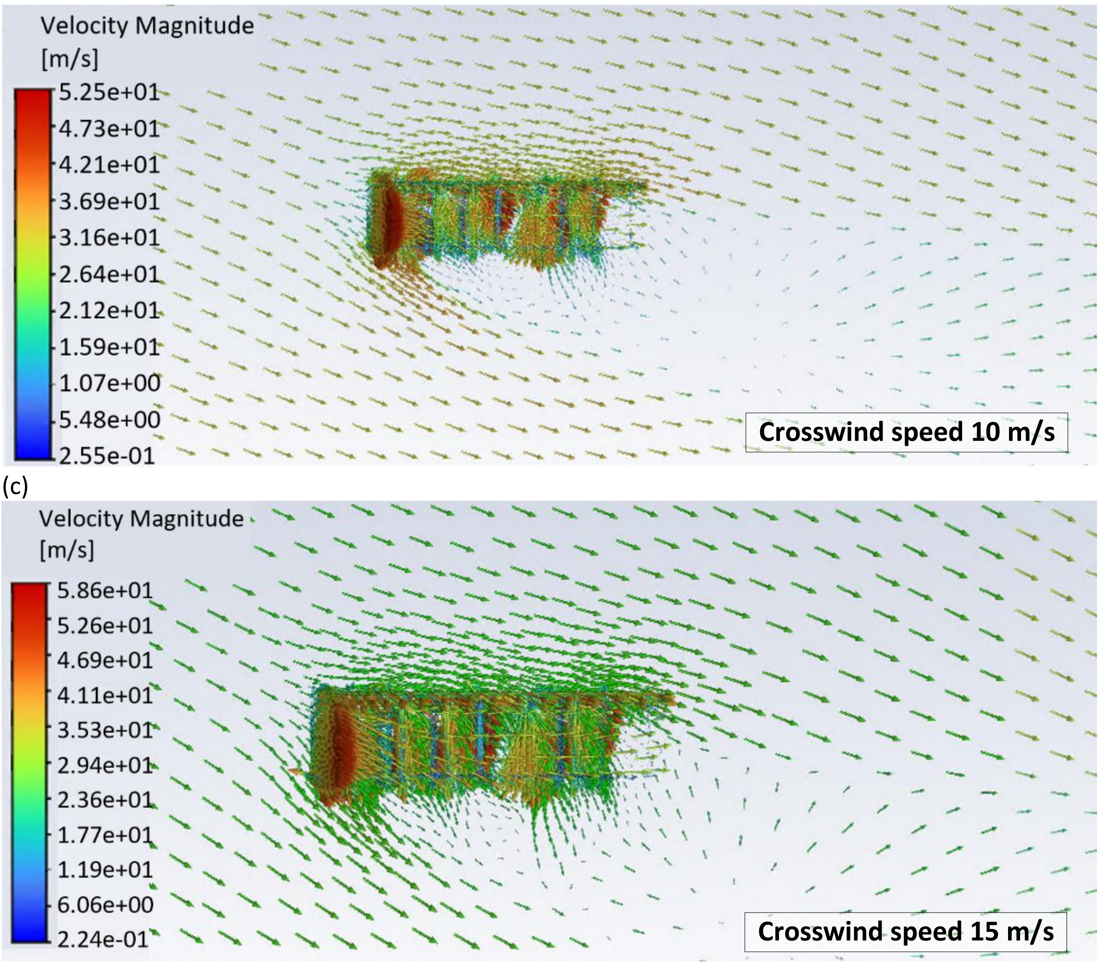
The primary aim of this study is to analyze airflow velocity vector distributions and evaluate fluid flow around the truck in both cases, identifying regions of high pressure, flow separation, and vortex formation at different crosswind intensities. Fig. 2 provides a top view of the induced airflow velocity vectors in the xz plane (representing the highway plane). In Fig. 2a, which shows the no-wind case, the typical airflow pattern around the truck is evident: a high-pressure zone forms at the truck's front where airflow slows, then deflects and accelerates along the truck's surface, creating symmetrical high-pressure, low-velocity pockets that contribute to vehicle drag.



(a)



(b)



(c)

(d)

Fig. 2. Distribution and magnitude of the induced airflow velocity vectors in the Phase 1, at crosswind speed of: a) 0 m/s, b) 5 m/s, c) 10 m/s, d) 15 m/s

As lateral wind velocity increases (Fig. 2bcd), the primary stagnation point remains at the truck's front, but the high-pressure region extends to the vehicle's side, which now acts as an obstacle. This produces an asymmetrical airflow distribution, increasing drag forces in both the x and z directions (Table 1).

Table 1
The values of Drag Coefficients at different crosswind velocities

Analyzed traffic phase	Crosswind velocity	Drag Coefficient	
		x Direction	z Direction
Phase 1: Truck	0 m/s	0.74	0.00
	5 m/s	1.04	0.15
	10 m/s	1.14	0.41
	15 m/s	1.17	0.61

At the truck's rear, flow separation occurs, forming a wake region. The sharp surface transition at the back leads to abrupt flow separation, resulting in significantly higher turbulent kinetic energy and Reynolds numbers in the expanded separation regions caused by increased lateral wind (Fig. 3abcd).

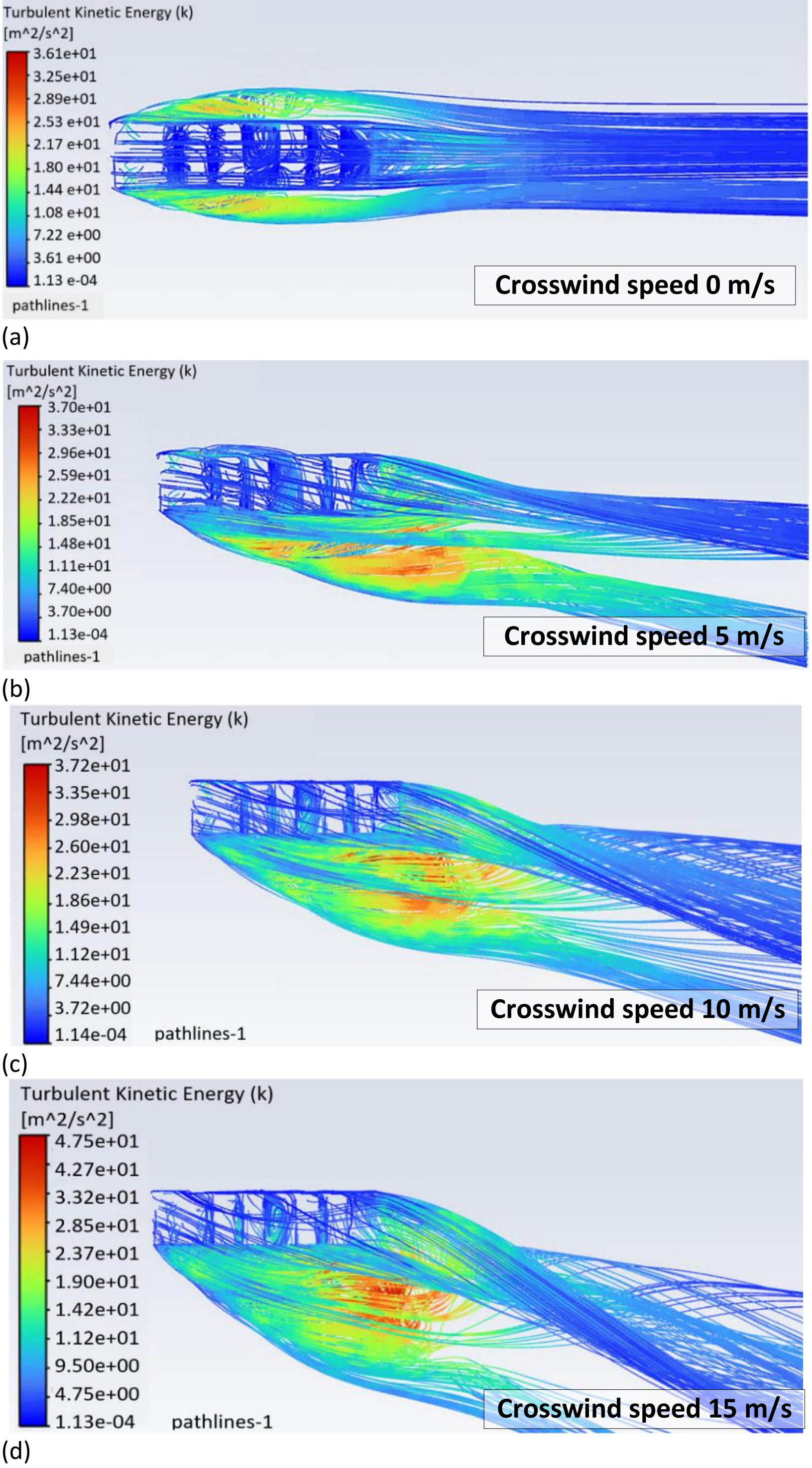
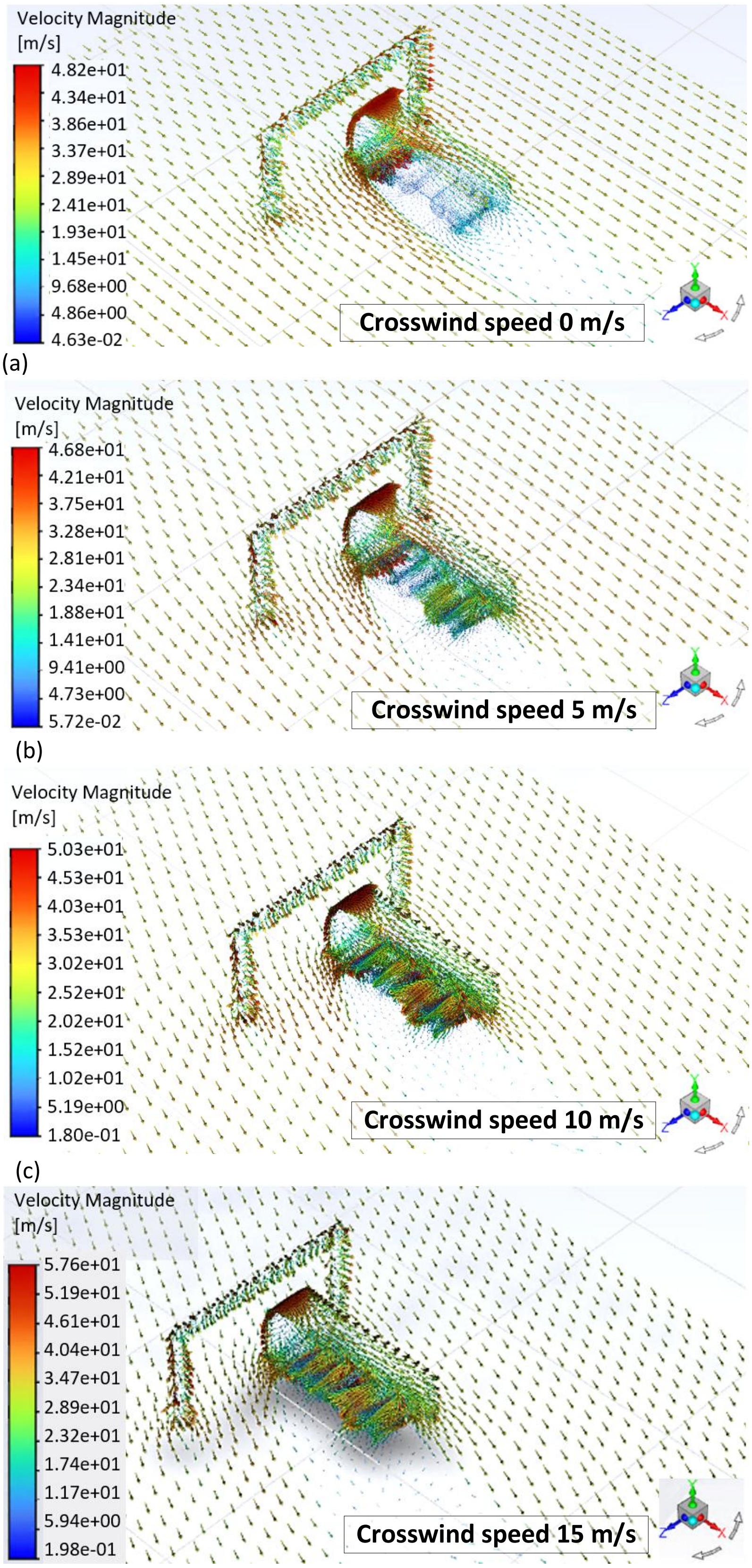


Fig. 3. Turbulent Kinetic Energy in Phase 1, at crosswind speed of:
(a) 0 m/s, (b) 5 m/s, (c) 10 m/s, (d) 15 m/s

In the second phase of our numerical analysis, where the truck passes beneath the portal, the velocity distribution in the absence of lateral wind reveals asymmetry on the truck side closer to the portal, Fig.4a, as the airflow perpendicular to the truck's front and the portal is channeled by these solid surfaces. At higher lateral wind speeds, the primary airflow strikes the side of the truck, increasing drag in the z direction by creating a high-pressure zone on the wind-exposed side, Fig.4bcd.



(a)

(b)

(c)

(d)

Fig. 4. Distribution and magnitude of the induced airflow velocity vectors in Phase 2, at different crosswind speed

This results in higher drag coefficient values (Table 2), indicating increased air resistance on the truck in the x direction. However, the presence of the portal pillar disrupts the lateral airflow, providing a partial shielding effect for the truck and reducing drag in the z direction in comparison with the values obtained in Phase 1.

Table 2
The values of Drag Coefficients at different crosswind velocities

Analyzed traffic phase	Crosswind velocity	Drag Coefficient	
		x Direction	z Direction
Phase 2: Truck & Portal	0 m/s	0.81	0.01
	5 m/s	1.04	0.13
	10 m/s	1.14	0.35
	15 m/s	1.17	0.53

The airflow effect on the portal-type highway sign support structure is represented by a pressure applied as distributed load across all its surfaces. This load, calculated in ANSYS Fluent and imported into Static Structural, reaches peak values of 768 Pa in the no-wind case and 917 Pa in the strongest lateral wind case (Fig.5).

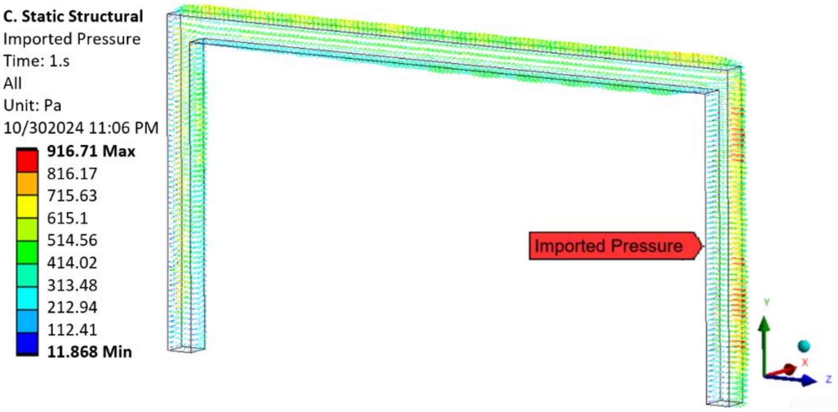


Fig. 5. The airflow effect on the portal: imported pressure in Phase 2 at crosswind speed of 15 m/s

The impact of the truck induced airflow and lateral wind on the portal, measured in terms of deformations, is minimal across all cases considered, with maximum values between 0.29 and 0.34 mm in the portal's upper section, primarily due to deformation in the x direction. For comparison, the maximum total deformation from the portal's own weight is approximately 1.8 mm. The maximum Von Mises stresses range from 1.07 to 1.42 MPa at the fixed support points of the portal pillars. While the immediate effects of airflow induced by truck movement and lateral wind on this type of steel portal appear negligible, it is essential to account for these forces in the design of lighter, less robust portal structures, as prolonged exposure to such disturbances may lead to significant damage over time.

4. Conclusions

In automotive engineering, accurately predicting and reducing air drag is essential for improving vehicle performance, fuel efficiency, and overall safety. This study contributes to understanding crosswind effects on large vehicles, as well as their impact on the road infrastructure. Our findings can be summarized as follows:

- Flow Distribution Analysis: The analysis shows that crosswinds significantly influence the flow distribution around the truck body. With increasing crosswind intensities and varying angles, regions of high pressure are observed on the windward side, leading to asymmetric flow around the vehicle. Flow separation occurs predominantly on the leeward side, where vortices form and intensify as crosswind angles increase. These vortices contribute to lateral forces and add complexity to the aerodynamic behavior, affecting overall vehicle stability.
- Drag Variation Analysis: The drag force experienced by the truck varies notably under different crosswind conditions, with higher drag observed at larger crosswind angles and intensities. These fluctuations can increase energy consumption and impact vehicle handling, especially at higher speeds. Quantifying these drag variations highlights the importance of crosswind considerations in aerodynamic design, as they can affect fuel efficiency and the vehicle's ability to maintain stability in real-world driving scenarios.
- Estimation of Truck-Induced Wind Gusts (TIWG): The study on TIWG reveals that gusts generated by the truck imposes additional forces on roadside infrastructure. These gusts interact with crosswinds to produce pressure and vibrational effects on nearby traffic signs and highway signal structures. The combined forces from TIWG and crosswinds suggest that current infrastructure design may need to incorporate wind-resistant features to withstand such pressures, especially in high crosswind areas. The insights gained here inform the design and placement of roadside structures to minimize risks posed by truck-induced gust forces.

The current analyses underscores the significant role crosswinds play in affecting truck aerodynamics, drag forces, and the impact on roadside infrastructure, pointing to the importance of comprehensive crosswind resilience strategies in both vehicle and infrastructure design.

Author Contributions

Conceptualization, D.A., A.M., M.B. and R.V.Z.; methodology, M.B. and R.V.Z.; formal analysis, M.B. and R.V.Z.; investigation, D.A., and A.M.; writing—original draft preparation, A.M., and M.B.; writing—review and editing, M.B. and R.V.Z.; visualization, D.A., and A.M. All authors have read and agreed to the published version of the manuscript.

Data Availability Statement

The data supporting the reported results was obtained through numerical simulations performed by the authors on the cluster of the Politehnica University Timisoara, Romania.

Conflicts of Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgement

This research was not funded by any grant.

References

[1] She, R., Ouyang, Y., Al-Qadi, I. L. (2018). CDF Analysis and Prediction Model for Air Resistance on Platooned Freight Trucks. Report No. ICT-20-011, Illinois Center for Transportation. https://doi.org/10.36501/0197-9191/20-011

[2] Favre, T. (2011). Aerodynamics simulations of ground vehicles in unsteady crosswind. Doctoral Thesis, KTH School of Engineering Sciences, ISSN 1651-7660. ISBN 978-91-7501-196-7. Available from: https://www.researchgate.net/publication/266892346_Aerodynamics_simulations_of_ground_vehicles_in_unsteady_crosswind [Accessed 30.10.2024]

[3] Zhu, H., 2021, Aerodynamic Analysis Of Utility Truck Safety In Severe Environments, all ETDs from UAB, 725. Available from: https://digitalcommons.library.uab.edu/etd-collection/725 [Accessed 04.11.2024]

[4] Al Shboul, K. W., Rasheed, A. H., Alshareef, A. H. (2021). Intelligent approach for accurately predicting fatigue damage in overhead highway sign structures. Structures 34, 3453-3463. https://doi.org/10.1016/j.istruc.2021.09.090

[5] Yang, S. C., King, J. P. C., Hong, H. P. (2020). Validation of fatigue design wind loads for natural wind gusts and for truck-induced wind gusts using full-scale measurements. Journal of Wind Engineering & Industrial Aerodynamics, 198, 104084. https://doi.org/10.1016/j.jweia.2019.104084

[6] Bojanowski, C., Lottes, S. A., Sitek, M. A. (2019). CFD Estimation of Truck Induced Wind Gusts on Variable Message Signs. Argonne National Laboratory. https://doi.org/10.2172/1546789

[7] Grm, A., & Batista, M. (2017). Vehicle Aerodynamic Stability Analysis under High Crosswinds. Strojniški vestnik - Journal of Mechanical Engineering, 63(3), 191-200. http://dx.doi.org/10.5545/sv-jme.2016.4095

[8] Brandt, A., Jacobson, B., Sebben, S. (2021). High speed driving stability of road vehicles under crosswinds: an aerodynamic and vehicle dynamic parametric sensitivity analysis. International Journal of Vehicle Mechanics and Mobility, 60 (7), 2334-2357. https://doi.org/10.1080/00423114.2021.1903516

[9] Backer, C. J. (1990). Ground vehicles in high cross winds. Part I: Steady aerodynamic forces. Journal of Fluids and Structures, 5, 69-90. https://doi.org/10.1016/0889-9746(91)80012-3

[10] Backer, C. J. (1991). Ground vehicles in high cross winds. Part II: Unsteady aerodynamic forces. Journal of Fluids and Structures, 5, 91-111. https://doi.org/10.1016/0889-9746(91)80013-4

[11] Backer, C. J. (1991). Ground vehicles in high cross winds. Part III: The interaction of aerodynamic forces and the vehicle system. Journal of Fluids and Structures, 5, 221-241. https://doi.org/10.1016/0889-9746(91)90478-8

[12] Stojanovic, N., Grujic, I., Boskovic, B. (2023). The influence of the crosswind on the lift coefficient, vehicle stability and safety. Annals of Faculty Engineering Hunedoara – International Journal of Engineering, XXI (4). Available from: https://annals.fih.upt.ro/pdf-full/2023/ANNALS-2023-4-11.pdf [Accessed 04.11.2024]

[13] Levin, J., Chen, S. H. (2022). Flow Structure Investigation of a Truck under Crosswinds. Journal of Applied Fluid Mechanics, 15 (5), 1513-1523. https://doi.org/10.47176/jafm.15.05.1076

[14] Kacin, J., Rizzo, P., Tajari, M. (2010). Fatigue analysis of overhead sign support structures. Engineering Structures, 32, 1659–1670. https://doi.org/10.1016/j.engstruct.2010.02.014

[15] Albert, M. N., Manuel, L., Frank, K. H., Wood, S. L., 2007, Field Testing of Cantilevered Traffic Signal Structures under Truck-Induced Gust Loads, CTR Technical Report. Available from: https://library.ctr.utexas.edu/ctr-publications/0-4586-2.pdf [Accessed 04.11.2024]

[16] Governing Equations of Fluid using Ansys Fluent. Available from: https://www.ansys.com/academic/educators/education-resources/teaching-package-governing-equations-of-fluids [Accessed 04.11.2024]

[17] ANSYS Evaluation of Gradients and Derivatives. Available from: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node368.htm [Accessed 04.11.2024]

[18] Menter, F. R., Lechner, R., Matyushenko, A., 2021, Best Practice: RANS Turbulence Modeling in Ansys CFD. Available from: https://www.ansys.com/content/dam/amp/2022/march/quick-request/Best%20Practice%20RANS%20Turbulence%20Modeling%20in%20Ansys%20CFD.pdf [Accessed 04.11.2024]

[19] ANSYS Pressure-Velocity Coupling. Available from: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node373.htm [Accessed 04.11.2024]

[20] 3D Models. Truck. Available from: https://3dmodels.org/3d-models/mercedes-benz-actros-tractor-3-axis-2011/ [Accessed 04.11.2024]