Shape Optimization Investigation of Next Generation Truck Design for Aerodynamic Efficiency

Faegheh Ghorbanishohrat, Brian R. McAuliffe

National Research Council Canada
Ottawa, Canada

Faegheh.ghorbanishohrat@nrc-cnrc.gc.ca
Brian.McAuliffe@nrc-cnrc.gc.ca

Abstract: Emerging zero-emission heavy-duty vehicle (ZEHDV) concepts provide a unique opportunity in the industry to optimize heavy-duty vehicle (HDVs) shapes with a view to increasing the aerodynamic efficiency of these vehicles. Accordingly, a research project was initiated to examine the potential energy savings and range extension of ZEHDVs associated with reduced aerodynamic drag. Preliminary experiments on a conceptual cab design with a modified front, including shallower surface angle and large corner radii, were performed in the NRC 9m Wind Tunnel using a 30% scale truck model. Results revealed a 7-9% reduction in the aerodynamic drag-area of the vehicle. For further study, Computational Fluid Dynamics (CFD), and parametric design were utilized to examine various cab geometries through airflow, surface pressure, and drag coefficient analyses. Various geometry cases were considered based on aerodynamic characteristics such as corner radii, cab frontal area, windshield angle, bumper shape, and underbody airflow. CFD results have shown up to 8% variability in drag coefficient resulting from changes to the shape of the cab model with these aerodynamic considerations. This investigation has guided the design of a new ZEHDV model for wind-tunnel testing in a subsequent phase of the study.

1 Introduction

Aerodynamic drag significantly affects the energy consumption of on-road vehicles, with heavy-duty tractor-trailer combinations using approximately half of their engine power to overcome this resistance at highway speeds [1,2]. As heavy-duty manufacturers transition to new power sources and drivetrain technologies, there are opportunities to enhance the aerodynamic performance of these vehicles. Some manufacturers are introducing radically redesigned zero-emission trucks, while others maintain traditional shapes. These changes can impact overall aerodynamics and alter the benefits of trailer aerodynamic technologies. It's essential to understand the differences in aerodynamic performance between conventional and emerging ZEHDV concepts to evaluate energy efficiency for various duty cycles, informing energy efficiency and regulatory initiatives for a decarbonized future in freight transportation [3,4].

This study addresses this subject by investigating aerodynamic shape optimization of emerging ZEHDVs through wind tunnel testing and computational fluid dynamics analysis.

2 Experimental Setup

Wind tunnel: The experiments were conducted in the NRC 9m wind tunnel, a closed-loop facility featuring a test section measuring 9.1m wide × 9.1m high × 22.9m long, with a 6.1m diameter turntable. The wind tunnel is equipped with a 6.7 MW DC motor that drives a fan capable of generating wind speeds of up to 55 m/s, and a ground effect simulation system. To measure aerodynamic forces and moments, a 30% scale model is affixed above a rolling road, supported by streamlined struts connected to a six-axis balance.

Test models*:* Experiments employed a 30% scale tractor-trailer model from NRC, mimicking diverse heavy-duty vehicle (HDV) setups. It offers flexibility for various cabin types and trailer variations, with the default configuration being a dry-van semi-trailer with and without aerodynamic devices. The three tractor variations are characterized as follows and displayed in Figure 1:

- The day-cab tractor: initially resembling an international ProStar short sleeper cab, the tractor underwent adjustments to the bumper, hood, A-pillars, and roof fairing. This adaptation enables it to transform into a more compact high-roof day-cab model, utilizing the same front components while incorporating unique cab-back and high-roof fairing designs by NRC, aimed at enhancing airflow efficiency over the trailer.
- The long-sleeper-cab tractor: is derived from a 3D scan of a 2016 Kenworth T680 high-roof long-sleeper cab. This model closely replicates the external shape of the full-scale vehicle, although some differences exist in engine bay, suspension, and drive-line components due to its underlying structure based on the short-sleeper model.

- The zero-emission-cab tractor: is derived from a ProStar day-cab tractor, but it has been altered with changes above the hood and in front of the windscreen to emulate the features seen in emerging ZEHDVs. These adjustments involve the use of a hand-carved modeling-foam insert to achieve a more gradual surface angle and incorporate larger corner radii. Additionally, the upper front grille was masked to replicate the reduced cooling air-flow [5].

Figure 1: Three wind tunnel tractor-trailer scale models: day-cab tractor(left), sleeper-cab tractor (middle), and zero-emission-cab tractor (right)

3 Computational Setup

CFD simulations were conducted using the Ansys-Fluent solver. A simplified 5% scale HDV model was initially used to conduct cross-validation with experimental data and identify the appropriate simulation process. The next step involved the optimization of the ZEHDV cab design through the parametric geometry and selected CFD simulation method [4].

3.1 Models

Preliminary HDV model*:* A 5% scale model of a simplified HDV contained shape characteristics of emerging ZEHDV concepts based on a research model developed at NRC. The cab length represents a model approximately halfway between a day-cab and a sleeper cab with an equivalent 40 ft trailer length [6,7], Figure 2.

Parametric Cab Design Model: For aerodynamic optimization of cab geometry while retaining the legal restrictions of heavy vehicles in North America, a parametric CAD model was created using SolidWorks software. Cab width, height, length, corner and roof radii, area of stagnation region, windshield and roof angles, wheel house shape, and height of under-body from the ground are geometry parameters that affect aerodynamic performance [8] and were considered in preparing the parametric geometry of the ZEHDV model.

The model was initially prepared at 30% scale to be comparable with the existing NRC long-sleeper-cab tractor model. For subsequent analyses, a full- scale version was utilized in CFD simulations. The clearance height from bumper to the ground was set to 0.024 m and the cooling flow was not represented in the tractor model. The trailer model was modified by adding a side-skirt with a chamfered front edge, representing flush-mounted skirts with full-scale dimensions of 7.9 m in length (26 ft) and 0.91 m in height (3 ft), and by simplifying the geometry of the trailer parts and removing design features that were considered to be insignificant. Figure 3 shows the initial design with large-radius corners and an overall shape similar to emerging ZEHDVs with no mirrors added to the geometry. This model was considered the ZEHDV baseline and was used for the first case study in the CFD aerodynamic optimization process.

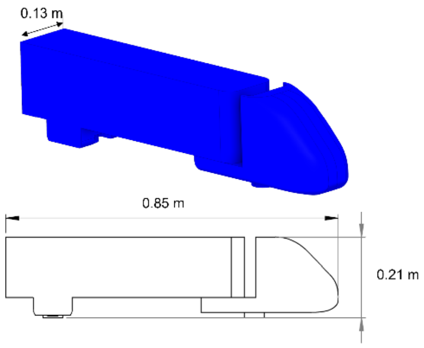

Figure 2: 5% scale preliminary HDV model

3.2 Simulation setup

Various turbulence models, such as Reynolds Averaged Navier-Stokes (RANS), Unsteady Reynolds Averaged Navier-Stokes (URANS), Detached Eddy Simulation (DES), and Delayed Detached Eddy Simulation (DDES), were assessed at different yaw angles (0° and 15°) using the 5% scale model and its available experimental data. Although not an exact match, the results showed similarities in flow behavior and the impact of yaw angle on forces. The study employed different turbulence models, poly-hexcore meshes, and boundary conditions based on NRC wind tunnel experiments. The RANS approach with the k-ω SST model was selected for the CFD optimization process, requiring more than 25 updates to achieve the optimized aerodynamic geometry of the ZEHDV model.

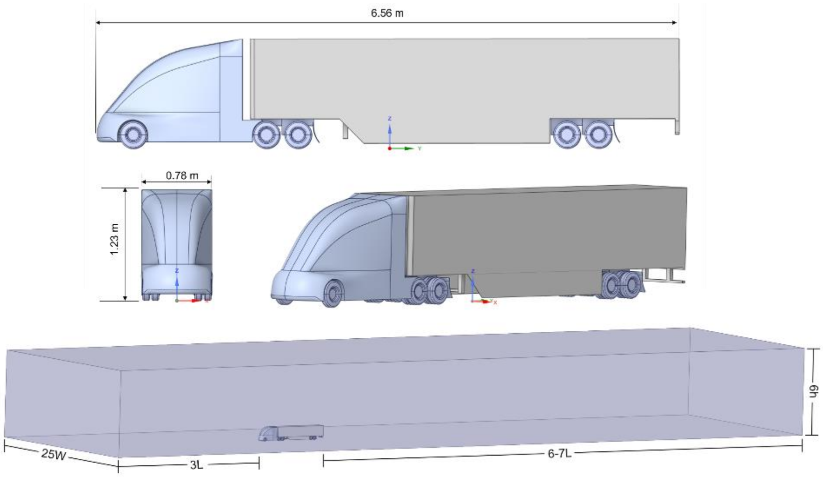

Figure 3: Schematic of the 30% scale model of the baseline aerodynamic model the computational domain

The computational domain was defined based on a literature review and industry best practices for simulating commercial ground vehicles [9,10,11], featuring a length of 242 m, extending 70 m ahead of and 150 m behind the vehicle, with the width of 64 m, and a height of 26 m for ZEHDV model. This represents a domain that extends approximately 3L upstream, 7L downstream, 25W wide, and 6H high according to the vehicle dimensions (L×W×H). The width of the domain was increased for cases where the yaw angle was above 7° to maintain the required clearance distance from the side walls. The domain was divided into multiple bodies of influence to refine the mesh size in regions of interest, such as the curvature around the tractor model and behind the trailer, where the variation of flow properties is significant and all gradients need to be captured. The EPA methodology of using a single yaw angle as a surrogate wind-averaged-drag metric was applied and all ZEHDV cases were simulated at yaw angle of 4.5°, with a speed set to 29 m/s.

Poly-hexcore meshes, which contain polyhedral and hexcore cells, were used for the meshing. In the full-scale ZEHDV simulations, this meshing provides about 16 million cells and 30 million nodes, with a minimum face area of 0.04 mm^2 and a minimum volume of 0.95 mm^3 specified in regions of high curvature around the tractor model. For regions requiring a lower mesh density, generally at greater distances from the model surfaces, the mesh density is coarsened. To simulate open road conditions for ZEHDV model, moving ground and rotating wheels boundary conditions were applied. Expanding the computational domain size, following SAE standards recommendations [12], reduced blockage, improved surface pressure accuracy in the third decimal point, with only a 1% difference in drag coefficient, affirming the adequacy of the smaller domain for simulations. In addition, the RANS simulations were repeated for a few ZEHDV cases, and each time, the drag coefficient was found to be within a 1% uncertainty range.

4 Results and Discussion

4.1 Experimental Results

The zero-emission-cab tractor was evaluated alongside three different dry-van semi-trailer configurations including standard trailer, trailer with skirts, and trailer with skirts and tail. Results were compared to the performance of both day-cab and sleeper-cab tractor models using the same three trailer setups. Figure 4 illustrates that the zero-emission-cab and sleeper-cab tractors share lower drag coefficients compared to the day-cab tractor, with sleeper-cabs benefiting from their elongated shape for improved aerodynamics.

The findings from the prior investigation involving the NRC ProStar-based sleeper-cab tractor indicate that cooling drag plays an insignificant role in the total drag. Consequently, the observed drag reduction is attributed to the streamlined design approach employed during its development, which involves strategies such as minimizing perpendicular or nearly perpendicular surfaces to the wind, particularly in the windshield area, and enhancing the curvature of front edges to extend surfaces with low-pressure forward.

A noteworthy finding is the contrast between the zero-emission-cab and sleeper-cab designs when paired with various trailer setups. They display similar drag coefficients, albeit with minor yaw asymmetry, in the case of the standard trailer. However, when equipped with side-skirts and a tail, the zero-emission cab demonstrates lower aerodynamic drag compared to the sleeper-cab, especially at lower yaw angles (approximately ±5∘), suggesting that alterations in the tractor's shape influence the effectiveness of drag reduction measures from trailer add-on devices such as side-skirts and a boat-tail [3].

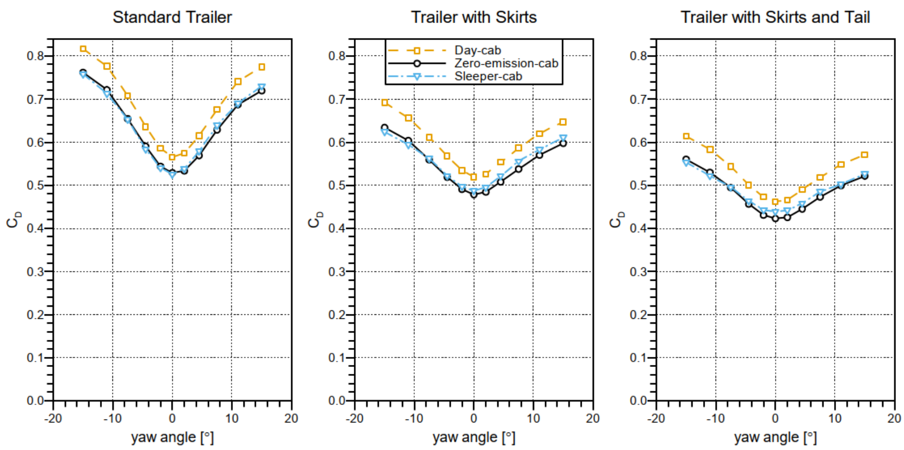

Figure 4: Measured variation of drag coefficient with yaw angle for the three tractor-model configurations paired with the three dry-van-semi-trailer-model configurations [3].

4.2 CFD Results

The preliminary experimental study demonstrated that an aerodynamic design could improve drag performance by up to 7-9% compared with a conventional day-cab. To further investigate the potential benefits of innovative cab designs, a CFD study was undertaken to create an optimized aerodynamic ZEHDV cab for further experimental investigation. CFD was used as a cost-efficient method to conduct parametric optimization that would not be feasible with experimental methods.

The study of RANS, URANS, and DES methods for the simplified CFD model showed that RANS simulations tend to underestimate the drag coefficient, but the trend of drag contribution for the tractor portion remained unchanged in RANS and transient solutions. Therefore, the differences in drag associated with shape changes are expected to track appropriately.

Accordingly, aerodynamic shape optimization simulations were performed using the RANS method with the $k - \omega$ SST turbulence model. A total of six design parameters were studied through 26 simulation cases, including: primary edge radii, bumper curve, curvature and angles of wind shield/cab roof, front surface area of the cab, length and width of the cab, air dams and under-body flow control. The objective was to focus on a specific design parameter in each iteration of geometry modification while preserving the overall shape. Although changing one design parameter can affect the overall parametric cab design, it was important to maintain the general shape. Additionally, in subsonic flows, any changes can affect the entire flow field, so modifications to some design parameters were repeated a few times in the optimization process.

Influence of Corner Radii and Cab's Front Area: Sharp corner edges can cause flow separation and increase a vehicle's drag. Using a proper curvature profile instead of sharp corners can create a smooth flow transition at the corners of the cab and prevent separation of the flow from the side panels of the cab. Multiple configurations were defined to optimize the corner radii via the parametrically designed geometry. Results showed that increasing the corner radius at the top corner created a high-pressure region, which led to the increased drag, while a slight decrease in the corner radius at the top of the tractor (a gradual decrease in corner radii from top to bottom) reduced the pressure and improved the overall drag coefficient. Additionally, the narrower front area showed the possibility of drag improvement by decreasing the surfaces experiencing the highest pressure. However, decreasing the front area exposes the side panels to the wind and results in an increased surface area perpendicular to the vehicle motion direction. Therefore, the surface experiencing the highest pressure decreased but created more high-pressure regions on the aft side surfaces of the cab, resulting in an increase in drag. Figure 5 shows three case studies investigating the effect of the front area and corner edge radius on surface pressure coefficient distribution and on drag coefficient accumulation. In comparison to the first geometry at left, second geometry (centre) resulted in less than 1% improvement, and the third geometry showed about 3% deterioration in aerodynamic drag coefficient.

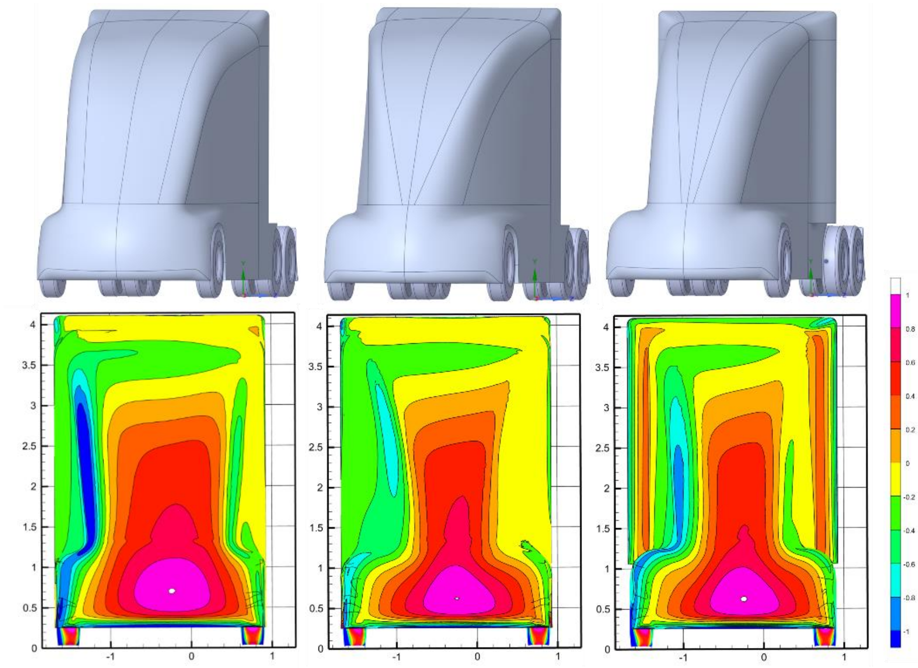

Figure 5: Surface pressure coefficient distribution; initial geometry (left), smaller corner radius (centre), narrower front area (right).

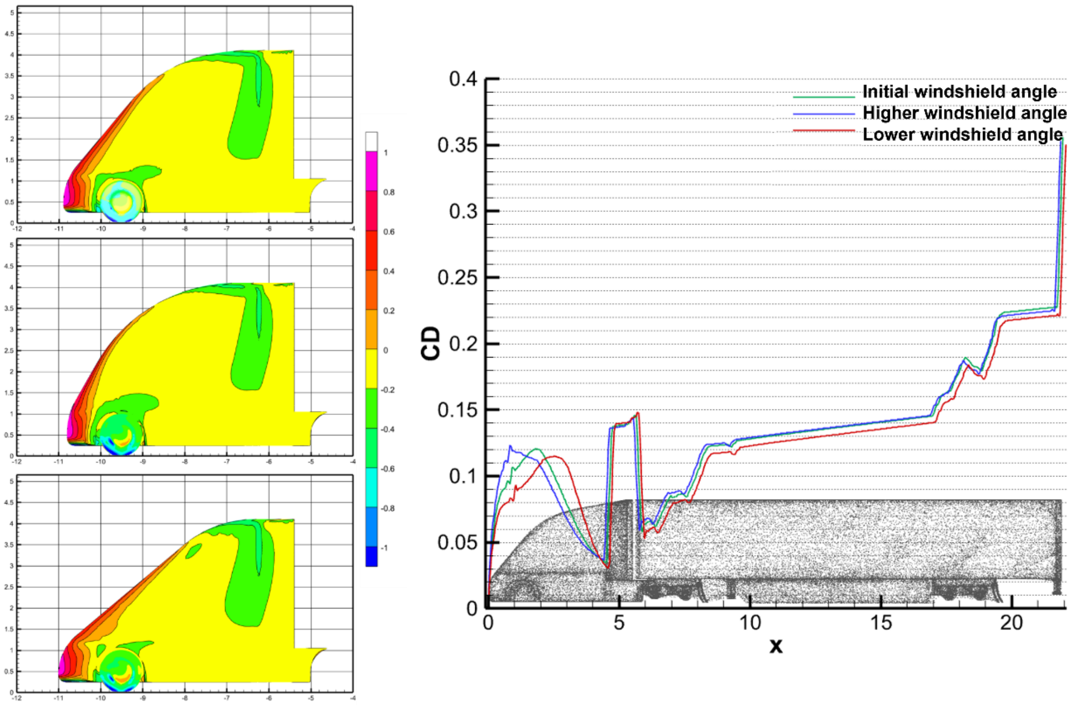

Figure 6: Influence of windshield and cab roof radius angle

Cab Roof Angle and wheelhouse, Increasing the cab height to reach the zero-cab roof angle added a 10.6% increase in the drag contribution of the front part of the cab. However, this modification decreased the drag contribution over the tractor-trailer gap region by more than 38%. Additionally, to guide the flow and prevent air from entering the wheelhouse, the front of the cab was slightly widened. Combining these modifications resulted in a 2.6% decrease in drag compared to the middle geometry in Figure 5.

Influence of Windshield and Cab Roof Radius Angle: the roof shape, height, and angle can guide the flow over the tractor-trailer gap and affect top edge vortex creation. Although the windshield angle and roof radius are dependent on each other, the effect of the windshield angle was studied by changing it by ±10°. Either increasing or decreasing the windshield angle by 10° decreased improved the aerodynamic drag up to 1.6%. This effect is precisely illustrated in the drag accumulation graph (Figure 6) which demonstrates that increasing the windshield angle moved the greater drag contribution toward the front of the cab by increasing the area of the perpendicular surface to the vehicle motion direction. On the other hand, a lower windshield angle lowered the front pressure drag, increased the suction region over the cab's roof, and also guided the flow along the cab surface, which reduced the contribution of the tractor-trailer's under-body drag.

Air dams can influence the underbody flow rate and considerably affect the aerodynamic performance of HDVs. The drag contribution of the underbody depends on many factors, such as the roughness, rate, and turbulence features of the underbody flow. The best way to mitigate underbody drag contribution, despite the turbulence intensity and underbody roughness, is to decrease the underbody flow rate as much as possible. This approach can be seen in most modern trucks, with flexible air dams extending almost to the ground. Shape and location of air dams were studied in multiple configurations. Adding air dams resulted in an increase of up to 2.3% or a decrease of as much as 1.6% in aerodynamic drag, depending on the configurations.

Individual design parameters were studied through 22 cases. Subsequently, a combination of the beneficial changes in design parameters was applied since, in subsonic flows, any changes affect the entire flow field, which can change the relative effectiveness of individually optimized parameters. Figure 7 shows the final geometry concept with increased windshield angle, improved curvature in the bumper's corner and three air dams including wheel and center ones selected from the CFD optimization process.

The current NRC 30%-scale model has an engine bay with a representative static engine and an electric-wheel-drive system. The new aerodynamically-optimized model would require some changes to the chassis, particularly a reduction in the size of the structure surrounding the engine bay, to provide room for the narrower cab shape at the front. However, the simulated cooling package, which defines the frontal area of this structure, will be needed for the increased cooling-flow necessary for a future fuel-cell variant of the new cab model. Therefore, to prevent any chassis changes, the geometry was slightly adjusted to envelop the chassis without interference. The final geometry concept and the current state of the physical 30% scale model, that is being designed from this concept, are shown in Figure 7.

Figure 7: Optimized Aerodynamic model (left), New designed physical 30% scale model (right)

5 Conclusions

The initial experiment demonstrated a 7-9% drag performance improvement with an aerodynamic design over a standard day-cab, prompting a CFD study to optimize an aerodynamic ZEHDV cab. Therefore, RANS simulations and parametric design were utilized to examine various cab geometries. Results showed larger cab corner radii and a gradual decrease in top corner radii improved aerodynamic performance by guiding the flow and mitigating edge vortices. A lower windshield angle reduced front pressure drag, while a flat zero cab roof with a larger roof radius created a larger suction region and improved performance. As expected, decreasing the surface area perpendicular to the vehicle direction of motion helped reduce drag, but decreasing the frontal area of the cab, while flaring its aft surfaces to match the trailer, should be approached with caution. To improve aerodynamics, it was important to have smooth curvature connections and prevent air from entering regions such as the wheelhouse. A smaller bumper radius with a smooth under body, minimum-length wheel air dams, and moving the central air dam further downstream all helped reduce the drag coefficient.

An optimized cab was achieved by studying 26 cases representing various geometries. The CFD results have shown up to 8% variability in drag coefficient resulting from changes to the shape of the cab model with aerodynamic considerations.

6 Acknowledgements

This research co-funded by both Transport Canada's ecoTECHNOLOGY for Vehicles program and the National Research Council Canada's Clean and Energy Efficient Transportation program. The views and opinions expressed by the authors do not necessarily state or represent those of Transport Canada.

7 Bibliography

[1] National Academy of Sciences, "Technologies and Approaches to Reducing the Fuel Consumption of Medium- and Heavy-Duty Vehicles" Committee to Assess Fuel Economy Technologies for Medium- and Heavy-Duty Vehicles, DOI: 10.17226/12845, National Academy of Sciences, 2010.

[2] Patten, J., McAuliffe, B. R., Mayda, W. and Tanguay, B. "Review of Aerodynamic Drag Reduction Devices for Heavy Trucks and Buses" Report No. CSTT-HVC-TR-205, 2012.

[3] McAuliffe, B. R., Ghorbanishohrat, F. and Barber, H. "Preliminary Investigation Towards Next Generation Truck Design for Aerodynamic Efficiency" NRC Report No. LTR-AL-2022-0069, National Research Council Canada, 2022.

[4] Ghorbanishohrat, F., McAuliffe, B. R.,"Shape Optimization Investigation of Next Generation Truck Design for Aerodynamic Efficiency" NRC Report No. AST-2023-0018, National Research Council Canada, 2023.

[5] McAuliffe, B.R., Hali, B., Ghorbanishohrat, F. "The Influence of Traffic Wakes on the Aerodynamic Performance of Heavy Duty Vehicles" SAE Paper 2023-01-0919, 2023.

[6] McAuliffe, B. R., Desouza, F. and Leuschen, J. "Aerodynamic Testing of Drag Reduction Technologies for HDVs: Progress Towards the Development of a Flow Treatment System" NRC Report No. LTR-AL-2012-0020, National Research Council Canada, 2013.

[7] McAuliffe, B. R., D'Auteuil, A. and de Souza, F. "Aerodynamic testing of drag reduction technologies for HDVs: progress toward the development of a flow treatment system (year 2)" NRC Report No. LTR-AL-2014-0014P, National Research Council Canada, 2016.

[8] Dillmann, A. & Orellano, A. "The Aerodynamics of Heavy Vehicles III: Trucks, Buses and Trains" Springer, 2015.

[9] "Guidelines for Aerodynamic Assessment of Medium and Heavy Commercial Ground Vehicles Using Computational Fluid Dynamics" SAE, 2013.

[10] Hoque, A. Z. U., Islam, M. A. and Shuvo, M. A. H. K. "Aerodynamic Shape Optimization of Vehicles Using CFD Simulation" International Conference on Mechanical, Industrial and Energy Engineering, 2018.

[11] Peng, J., Wang, T., Yang, T., Sun, X. and Li, G. "Research on the Aerodynamic Characteristics of Tractor-Trailers with a Parametric Cab Design" Applied Sciences, 8, 2018.

[12] "Guidelines for Aerodynamic Assessment of Medium and Heavy Commercial Ground Vehicles Using Computational Fluid Dynamics" SAE, 2021.