

CFD Simulation of Aerodynamic Forces on a Simplified Heavy Vehicle Model with Steady Base Blowing

Erhan Fırat^{1*}  and Mehmet Seyhan² 

¹Mechanical Engineering Department, Faculty of Engineering, Munzur University, Tunceli, 62000, Türkiye

²Mechanical Engineering Department, Faculty of Engineering, Karadeniz Technical University, Trabzon, 61080, Türkiye

Abstract

A numerical investigation was conducted to examine the flow around a European tractor-trailer combination model (GETS) using a steady RANS model $k-\omega$ SST at a Reynolds number of 100000. Experimental aerodynamic force measurements were also carried out for the purpose of validating the CFD simulations. The present paper focuses on the effect of steady blowing from the thin slots positioned on the rear surface (base) of the model on the aerodynamic characteristics of the GETS. The combination of three key parameters was tested: the jet-to-freestream velocity ratio (VR), the inclination angle of the jet (θ), and the jet's location. More than 100 CFD simulations were performed over a range of key parameters. The findings indicate that the jet direction's inclination towards the rear surface typically leads to a modest yet favorable effect on the drag reduction. It has been demonstrated that increasing the VR to 2 has the potential to reduce aerodynamic drag by up to 50%. However, it is also demonstrated that employing VR values greater than approximately 1.25 is not viable in terms of active flow control efficiency. At $\theta=0^\circ$, the active control efficiency of the both-jets configuration is approximately 6 for $VR=0.5$ and approximately 1.6 for $VR=1$. The total drag coefficient on the model is reduced by 8.3% and 17.7% for the $VR=0.5$ and 1 cases, respectively. It has been demonstrated that the viscous components of total drag and lift are independent of jet configuration, VR , and θ . In both upper and lower jet configurations, it is possible to increase or decrease the aerodynamic lift acting on the model by changing the inclination angle.

Keywords: Active flow control; Blowing; Computational Fluid Dynamics; Drag reduction; Planar steady jet; Truck-trailer

Research Article

History

Received 01.05.2026
Revised 28.05.2026
Accepted 09.06.2026

Contact

* Corresponding author
Erhan Fırat
efirat@munzur.edu.tr
Address: Mechanical
Engineering Department,
Faculty of Engineering,
Munzur University, Tunceli,
Türkiye
Tel: +905555104149

To cite this paper: Fırat, E., Seyhan, M. CFD Simulation of Aerodynamic Forces on a Simplified Heavy Vehicle Model with Steady Base Blowing. International Journal of Automotive Science and Technology. 2026; 10 (2): 411-426. <https://doi.org/10.30939/ijastech..1941954>

1. Introduction

It is reported that heavy-duty vehicles (HDVs), including lorries with a maximum permissible laden mass of over 3.5 tones, as well as buses and coaches, are currently responsible for over a quarter of greenhouse gas emissions from road transport in the European Union (EU) and over 6% of the EU's total GHG emissions [1]. In order to achieve the EU's goal of climate neutrality by 2050 (which requires a 90% reduction in transport emissions), the EU has introduced a number of new legislative measures in recent years to encourage improvements in the CO₂ emissions performance of HDVs.

One method for tractor and trailer manufacturers to reduce the CO₂ emissions and fuel consumption of HDVs is to improve their aerodynamic performance. The enhancement of a vehicle's aerodynamic performance can be achieved through the implementation of a flow control method on the concerned vehicle.

The classification of flow control methods is typically into two main categories: active and passive. The former methods (i.e. active) necessitate the utilization of external energy, whereas the latter do not. In the study by Hariram et al. [2], a range of passive aerodynamic solutions for European tractor and semi-trailer configurations were presented and discussed. These solutions were then summarized in terms of their impact on aerodynamic drag, and the potential contribution of this to the reduction of fuel consumption. McAuliffe [3] employed a 30% scale model of modern tractor-trailer combinations to investigate the aerodynamic improvements possible with current and emerging passive drag reduction technologies for heavy-duty vehicles, with the intent of guiding future implementation and regulation of such technologies for Canada's transportation industry. As outlined in the comprehensive report, the wind-tunnel measurements were utilized for two primary purposes: firstly, to estimate a drag coefficient for typical North-American wind conditions,

and secondly, to estimate fuel savings and greenhouse-gas reductions based on typical Canadian driving distances.

As is the case with the majority of extant studies, the aforementioned works have primarily centered on passive flow control methods for minimizing aerodynamic drag in the context of the tractor-trailer configuration [2-8]. The primary rationale for this is that they can substantially minimize the drag affecting the tractor-trailer configuration without necessitating external energy sources or complex electronic circuits. Conversely, active flow control methods offer significant advantages due to their capacity to be activated and deactivated as required and to adapt to changing environmental conditions. Consequently, a limited number of researchers have examined the potential of active flow control methods in reducing the drag coefficient of a tractor-trailer configuration. For instance, Phan et al. [9] performed numerical studies to minimize the aerodynamic drag on a 1:20 scale tractor-trailer combination using pulsed jets. The jets were ejected from thin slots near the edges of the rear surface of the trailer and directed toward the base region. They used the SST $k-\omega$ turbulence model. They showed that, when optimized jet parameters were used, pulsed jet increased the base pressure. However, the numerical study did not provide the net reduction in time-averaged drag. Vernet et al. [10] conducted a series of wind tunnel measurements with the objective of assessing the drag reduction capability of dielectric barrier discharge plasma actuators under cross-wind conditions. The DBD actuators were positioned on the A-pillar of the leeward side of the 1:6 scale model tractor-trailer combination, and the drag force was measured using a wind-tunnel balance. According to the available data, a drag reduction of approximately 20% is achieved at a yaw angle of 9° . In addition, it has been documented that a decrease in drag is observed as the yaw angle approaches a value of 0° . Some of the other studies are related to controlling flow separation at the A-pillar of a truck cabin using synthetic jets; increasing base pressure at the rear end of a large truck-trailer configuration driving at highway speeds with a 1/4-cylinder-shaped add-on device mounted on the rear end that has an array of synchronized suction and oscillatory blowing (SaOB) actuators; and improving the aerodynamic performance of the vehicle with circulation control on the rear surface [11-14].

Continuous steady jet/blowing is also an active flow control method, and its potential to reduce aerodynamic drag affecting different types of vehicles or vehicle models (Windsor model, DrivAer, Ahmed body, commercial van model, Honda simplified body) has been tested by various researchers [15-21]. In numerous tests, the device associated with the relevant flow control method is typically situated on the rear section of the relevant vehicle. However, the drag reduction potential of this method was not assessed on the HDVs. The aspect ratio (height to width ratio) of cars and vans differs from that of HDVs. The aspect ratio affects the aerodynamics and near-wake flow characteristics of road vehicles [22-24]. This constitutes the primary impetus for conducting the present study.

In the present study, the effects of continuous steady jets issued from the thin slots placed along the width of upper, lower, and both edges on the aerodynamic characteristics of a simplified European tractor-trailer combination model (GETS) are examined. To this end, steady-state flow simulations were conducted. To validate these flow simulations around the GETS model, experimental measurements were also conducted. The present numerical study focuses on three key parameters: the jet velocity ratio (VR), the angle at which the jet exits the model (θ), and the location of the jet.

The paper's structure is as follows. The experimental and numerical setups are described in Section 2. The verification and validation of the numerical study is also provided in this section. The effects of key parameters on the aerodynamic characteristics of the vehicle model were provided and discussed in Section 3. The conclusion in Section 4 ends the paper.

2. Setups and Techniques

2.1. Test model and key parameters

The model utilized in the wind tunnel measurements is depicted in Fig. 1. This model is a 1:26 scale GETS, first introduced by Van Raemdonck and van Tooren [25]. The primary dimensions of the model are delineated in the figure [6]. The distance between the ground and the model was maintained at 21 mm, and the leading edges of the model were rounded to 20.8 mm.

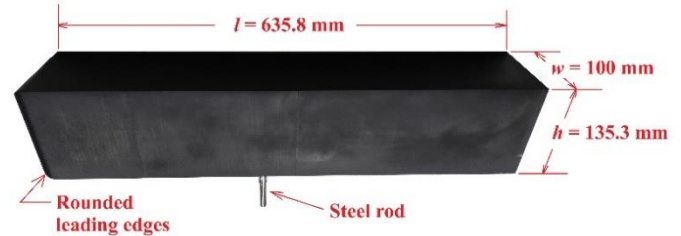


Figure 1. Dimensions of the GETS model installed in the test section of the wind tunnel.

Fig. 2 presents the key parameters and coordinate system utilized in the study. In the figure, V_{jet} denotes the jet velocity in m/s, and θ denotes the jet direction with respect to the main flow direction. The ratio of the V_{jet} to the freestream velocity (U_∞) is defined as the velocity ratio (VR). In the present numerical study, nine values of VR , ranging from 0 to 2 with 0.25 increments, were examined. It is evident that attaining elevated VR values for planar jets is only attainable through the implementation of a blowing system that necessitates additional spatial requirements and greater power consumption. Consequently, the upper limit was designated at 2. In other studies that employ planar jets to manage the flow around a road vehicle, the upper limit for this parameter has been set at a lower value [15, 19].

As the direction of the upper or lower jets inclines toward the rear surface of the model, the θ takes on a positive value (Fig. 2a). In other words, the inward inclination of the jet direction results in a positive θ value. In the present numerical study, nine

values of θ , ranging from -16° to $+16^\circ$ with an increment of 4° , were examined. As is evident from the extant literature, base flaps and boat tails are popular passive flow control devices [3-6]. These devices are utilized in order to reduce aerodynamic drag. As demonstrated in the extant literature, the optimal angle for the flat flap affixed to the base of the trailer is approximately $+13$ degrees [3, 4]. A planar jet generates an air curtain, which functions as a flat plate in terms of separating two flow regions. In the present study, the range of values for the θ was determined to encompass this optimal angle and to reveal the effects of both positive and negative angles on the aerodynamic characteristics.

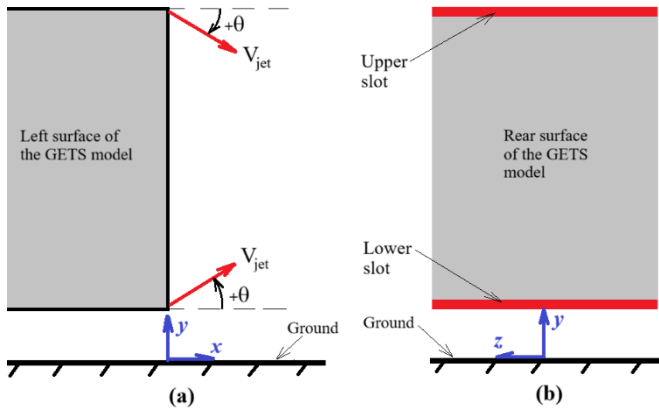


Figure 2. Schematic demonstration of the key parameters used in the study. a) Left view of the model b) rear view of the model. The adopted coordinate system is also delineated in the figure.

Lorite-Díez et al. [20] demonstrated that the optimal performance in terms of drag reduction is achieved by blowing from the slots located at the top, bottom, or both. The present study also adopts their slot locations. In the present numerical study, three jet configurations were examined: upper jet (i.e. jet is created only at the upper slot), lower jet (i.e. jet is created only at the lower slot), and both jets (i.e. jet is created concurrently on both slots). The locations of the thin slots, one on the top and the other on the bottom of the rear surface, are depicted in Fig. 2b. Both slots possess a thickness of 2 mm and extend along the width of the model. The utilization of slots with reduced thickness in the creation of a jet necessitates a lesser amount of energy and is also a more viable engineering approach [17-20].

The momentum coefficient is the dimensionless form of the force exerted by a jet flow, which is a direct consequence of the conservation of momentum. The momentum coefficient was calculated using the following formula.

$$C_\mu = \frac{\rho V_{jet} A_S (V_{jet} \cos \theta)}{0.5 \rho U_\infty^2 A_F} \quad (1)$$

where ρ denotes the air density at a standard sea level and ambient temperature of $22.5 \pm 1^\circ\text{C}$, A_S denotes the area of the slot, A_F denotes the frontal area of the GETS model.

2.2. Experimental setup

It is imperative to acknowledge that the present experimental study was conducted with the objective of validating the numerical study. The experiments described herein were conducted within the open circuit subsonic wind tunnel at Karadeniz Technical University (Fig. 3). The tunnel's contraction ratio is approximately 6.3:1. A variable frequency drive was integrated with the electrical motor of the axial fan to achieve the desired freestream velocity (U_∞) in the test section. The tunnel contains a 1200 mm long test section, with an inlet cross-section measuring 570 mm in width and 570 mm in height, and an outlet cross-section measuring 583 mm in width and 583 mm in height. The area-weighted freestream turbulence intensity (Tu_x) in the test section was approximately 0.7%.

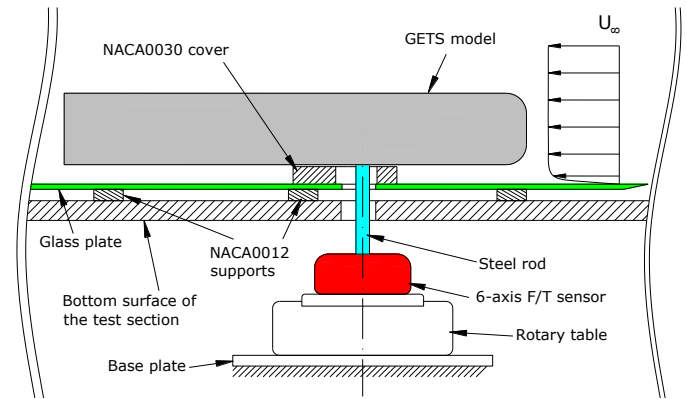


Figure 3. Sketch of the experimental set-up

The width-based Reynolds numbers of the experimental study ranged from $Re_w = 71000 \pm 1300$ ($U_\infty = 11$ m/s) to 130000 ± 3800 ($U_\infty = 20$ m/s). The width-based Reynolds number was calculated using the following formula.

$$Re_w = \frac{\rho U_\infty w}{\mu} \quad (2)$$

As indicated in the aforementioned equations, w denotes the width of the test model and μ denotes the viscosity of air. The GETS model was situated at the center of the bottom wall of the test section. The maximum effective blockage for the working section above the glass ground plate was approximately 4.25% in the no yaw case. It is noteworthy that no blockage correction was implemented, as the rate was found to be below the 5% threshold [26]. The model height was equivalent to 0.28 of the active test section height. The ratio of the clearance between the GETS model and the ground plate (g) to model width (w) was $0.210 \pm 2\%$. The 4 mm-thick ground plate was installed 12 mm above the inner surface of the bottom wall of the test section. This was done to reduce the thickness of the turbulent boundary layer. The thickness of the boundary layer on the plate was subsequently quantified through the utilization of a Pitot tube, which was connected to a digital differential pressure sensor (Sensirion, SDP810-125Pa). Velocity data were recorded at a

frequency of 250 Hz for a duration of 300 s. Boundary layer measurements were conducted on the ground plate in the absence of the GETS model at the station corresponding to the midpoint of the ground plate. The thickness of the boundary layer that has been measured to develop on the ground plate at the station is approximately 11 mm at $Re_w = 130000 \pm 3000$. The ground plate was extended $2.1w$ in front of the GETS model and $3.5w$ behind the GETS model, with w representing the width of the model.

The GETS model is connected by a steel rod with a diameter of 12 mm to a 6-axis force/torque (F/T) sensor (Schunk, FTD-Gamma SI-32-2.5; Fig. 2). The distance between the rod axis and the model base was approximately $3.45w$. In order to circumvent the necessity for corrections for tare forces and moments, the steel rod between the GETS model and the glass plate was encased within a hollow NACA0030-profiled part, with a chord length of 66.6 ± 0.2 mm. The height of this part is 19.0 ± 0.2 mm. Furthermore, it is imperative to note that this part is attached exclusively to the glass plate and does not come into contact with the GETS model or steel rod. The relatively diminutive vortical structures shed from the NACA0030 profile presumably dissipated before reaching the base of the GETS model, thereby exerting no effect on the flow characteristics in the GETS model wake. The F/T sensor was utilized to ascertain the time-averaged drag, F_D , and time-averaged lift, F_L , experienced by the GETS model. The signals from the F/T sensor were sampled at a rate of 2 kHz for a duration of 30 s. The time-averaged drag and lift coefficients, C_D and C_L , were calculated using the following equations, respectively.

$$C_D = \frac{2 F_D}{\rho U_\infty^2 A_F} \quad (3)$$

$$C_L = \frac{2 F_L}{\rho U_\infty^2 A_F} \quad (4)$$

The drag and lift coefficient calculations for the GETS model were accurate to within ± 0.02 . The width-based Reynolds number calculation was accurate to within $\pm 2.3\%$. The yaw angle measurement was accurate to within $\pm 0.1^\circ$.

2.3. Numerical setup

The computational domain geometry employed in numerical studies is designed to be identical to the interior of the wind tunnel test section utilized in experimental studies (Fig. 4). The sole distinction between the two models lies in the suspension of the GETS model employed in the numerical study. The height of the computational domain is equivalent to the vertical distance between the ground plate and the tunnel top wall utilized in experimental studies (i.e., 0.558 m). The width of the computational domain is equivalent to the width of the tunnel test domain (i.e., 0.570 m). The length of the computational domain is equivalent to eight times the length (l) of the GETS model. The GETS model is positioned at a height of 21 millimeters above the ground plate and at an equal distance from the side walls, as was established in the experimental study. The distance between the front surface of the model and the surface where the velocity

inlet boundary condition is assigned is equivalent to $2l$, while the distance between the rear surface of the model and the surface where the pressure outlet boundary condition is assigned is $5l$.

At the inlet boundary, the x -component of the velocity is the only component with a non-zero value that corresponds to the freestream value. Within this boundary, the free-stream turbulence intensity value is designated as 0.7%. At the outlet boundary condition, a value of 0-gauge pressure has been assigned to the entire surface. A no-slip wall boundary condition has been assigned to the model surface and tunnel walls.

The flow simulations were performed using the commercial finite volume solver ANSYS FLUENT (version 2020 R2). The steady formulation of the Reynolds-Averaged Navier-Stokes (RANS) equations was employed to solve the flow, despite the unsteady nature of the flow in the base of the GETS model. The rationale behind this choice is that extant literature has demonstrated the feasibility of accurately determining the aerodynamic forces acting on HDVs through the implementation of steady flow simulations [27-30]. Similar preferences are observed not only in external flow conditions but also in internal flow conditions [31]. The two-equation $k-\omega$ Shear Stress Transport (SST) turbulence model [32] was employed to solve for turbulence. The details of numerical formulation of the $k-\omega$ SST model can be found in [32].

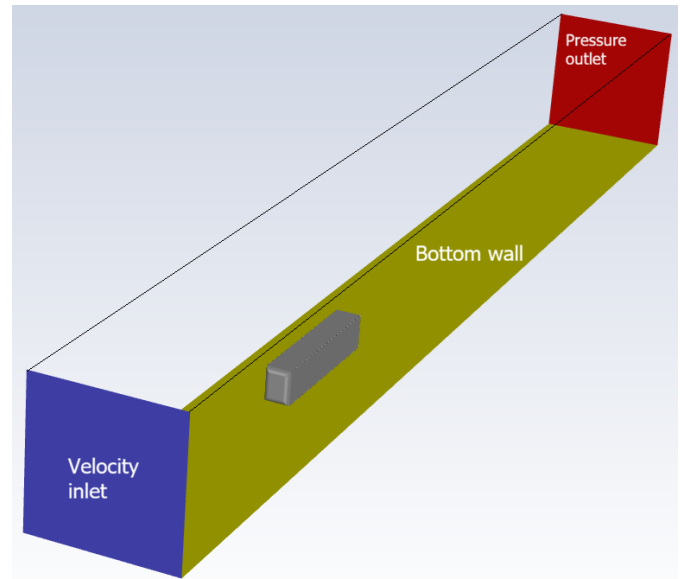


Figure 4. Geometry of the computational domain and boundary conditions. The GETS model is shown in gray within the figure. The side and top walls are transparent to ensure visibility of the interior of the computational domain geometry.

The program utilizes a control-volume-based technique to convert a scalar transport equation into an algebraic equation that can be solved numerically. Diffusion terms in the discretized scalar transport equation are central-differenced. The pressure-based solver was selected on the basis that the flow system under consideration did not exhibit compressibility effects. The gradients are

computed according to the Least Squares Cell-Based method. The velocity–pressure coupling is solved by means of the SIMPLEC scheme. The second-order scheme was utilized for the interpolation of pressure values at the interfaces between cells. The QUICK scheme was utilized to calculate a higher-order value of the any convected variable at a cell face. The turbulence quantities (turbulent kinetic energy and specific turbulent dissipation rate) were discretized using the first-order upwind scheme. The rationale underlying this phenomenon is that when the second-order upwind scheme was selected, the convergence criterion was not met. All simulations were executed until the continuity residual reached a value below 10^{-4} . The solver settings adopted in the present study are summarized in Table 1.

Table 1. Pressure-based solver settings

Turbulence model	k- ω SST
Production limiter	On
Curvature correction	On (CC=1.25)
Pressure-velocity coupling	SIMPLEC
Gradient scheme	Least squares cell based
Pressure discretization	Second order
Momentum discretization	QUICK
Turbulence discretization	First order
Residuals	0.0001

The multi-block analysis was adopted as a method for generating hexahedral cells in the computational domain (Fig. 5). In order to resolve the viscous sublayer, it is necessary to determine a value for the non-dimensionalized distance from the wall (y^+) that is approximately 1. The height of the wall adjacent mesh on the GETS model is around 0.03 mm yields a y^+ value of around 0.52, which is less than 1.

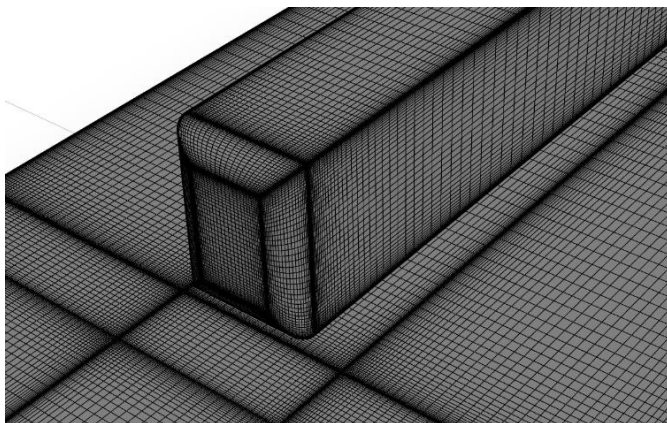


Figure 5. Visualization of surface meshes on the GETS model and on the ground plate. Block-structured hexahedral meshes were generated within the computational domain.

Table 2. Mesh dependency study conducted between four mesh cases. The variation of the GETS model drag and lift coefficients as a function of the total number of meshes.

Mesh size	Total mesh count	C_D	C_L
Coarse	3063648	0.387	-0.127
Medium	4553184	0.382	-0.119
Fine	6195104	0.379	-0.115
Finer	8256000	0.378	-0.115

Table 3. Validation study for the GETS model with fine mesh. The effects of curvature correction options on the results were also examined. Here, CC represents the specific limiter for the curvature correction function.

Method	Curvature Correction	C_D	C_L
Num. (SST k- ω)	Off (CC=0)	0.379	-0.115
Num. (SST k- ω)	On (CC=1.00)	0.382	-0.112
Num. (SST k- ω)	On (CC=1.25)	0.384	-0.111
Exp. (Direct force measurement)	-	0.436 $\pm$ 0.015	-0.0985 $\pm$ 0.0083

A mesh independence test was performed to determine the total number of meshes required for conducting the proposed flow simulation. To this end, volume meshes with four different levels of refinement were generated (Table 2). The results demonstrated that the percentage change in the drag coefficient is -0.26% as the mesh size is changed from fine to finer (Table 2). Additionally, the value of the lift coefficient remained constant at -0.115. Consequently, the fine mesh was selected for the subsequent simulations. In addition to conducting the mesh independence test, the effect of curvature correction on the aerodynamic characteristics was examined. This was necessary because the GETS model has rounded leading edges (Table 3) and the adopted turbulence model is insensitive to streamline curvature. The findings revealed that the values of both aerodynamic characteristics converged towards the experimental values as curvature correction was enabled. The drag and lift coefficients computed using the SST k- ω turbulence model with a curvature correction option (CC=1.25) are found to be -11.9% and -12.7% lower than those obtained from wind tunnel experiments. Consequently, the curvature correction option with a specific limiter of CC=1.25 was employed for the remaining simulations.

The drag and lift coefficients, calculated from direct force measurements and those obtained from the current numerical study, are compared in Fig. 6 for a specific Reynolds number range. The figure indicates that the experimentally obtained lift coefficient is nearly equivalent to the numerically obtained value. With regard to the drag coefficient, the numerical data exhibit a slight discrepancy from the experimental data. The observed discrepancies between the experimental and numerical studies can be primarily ascribed to three factors. Firstly, the utilization of a steady-state solver in simulating the flow around the model. Secondly, the accuracy of the turbulence model employed in solving the pertinent

problem. Thirdly, the disparity in boundary layer developments between the experimental and numerical studies.

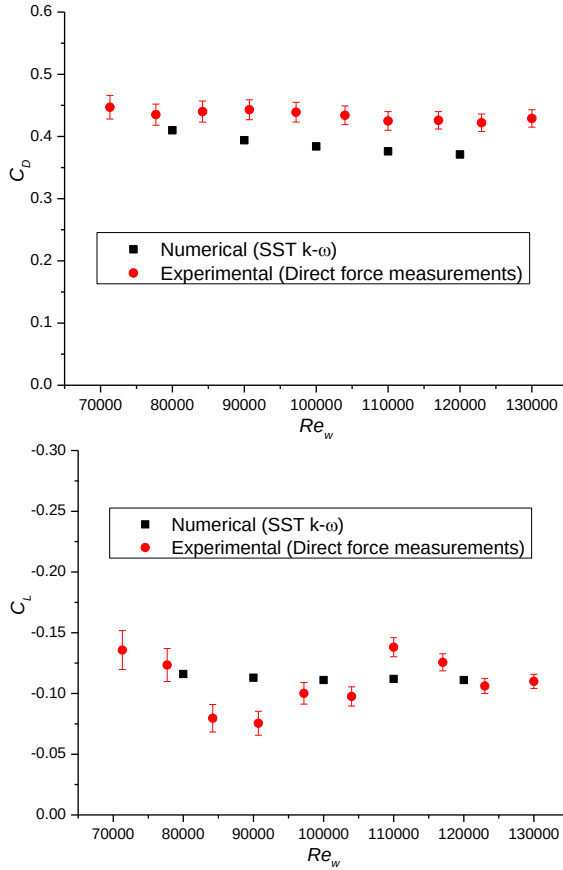


Figure 6. Validation of aerodynamic coefficients acting on the GETS model for different Reynolds numbers

In the present study, solely the geometric similarity was achieved. Additionally, it is not possible to match the GETS model Reynolds number to that of the prototype with this wind tunnel facility. In the extant literature, the majority of studies on road vehicles have not achieved complete similarity [5, 6, 9, 13, 19, 20]. However, it has been demonstrated that the drag reduction performance of a flow control device obtained in the case of incomplete similarity exhibits the same trends in the case of complete similarity [33]. In this study, values approaching those that could be obtained using the proposed flow control method on the prototype (full-scale model) will be obtained.

3. Results and Discussion

3.1. Aerodynamic coefficients

Prior to delving into this section, it is imperative to elucidate the role of the jet flow from the rear slots in the calculation of the aerodynamic characteristics. The momentum coefficient, which is the dimensionless form of the forces associated with the jet flow, was derived by utilizing the conservation of linear momentum (see Eq. (1)). The total drag and total lift coefficients

(i.e., C_{DT} and C_{LT}) which encompass the momentum coefficient, can be expressed as follows.

$$C_{DT} = C_{D-pre} + C_{D-vis} + C_{D-mom} \quad (5)$$

$$C_{DT} = \frac{F_{D-pre}}{0.5 \rho U_{\infty}^2 A_F} + \frac{F_{D-vis}}{0.5 \rho U_{\infty}^2 A_F} + \frac{\rho V_{u-jet} A_S (-V_{u-jet} \cos \theta)}{0.5 \rho U_{\infty}^2 A_F} + \frac{\rho V_{l-jet} A_S (-V_{l-jet} \cos \theta)}{0.5 \rho U_{\infty}^2 A_F} \quad (6)$$

$$C_{LT} = C_{L-pre} + C_{L-vis} + C_{L-mom} \quad (7)$$

$$C_{LT} = \frac{F_{L-pre}}{0.5 \rho U_{\infty}^2 A_F} + \frac{F_{L-vis}}{0.5 \rho U_{\infty}^2 A_F} + \frac{\rho V_{u-jet} A_S (V_{u-jet} \sin \theta)}{0.5 \rho U_{\infty}^2 A_F} + \frac{\rho V_{l-jet} A_S (V_{l-jet} \sin(-\theta))}{0.5 \rho U_{\infty}^2 A_F} \quad (8)$$

where F_{D-pre} is the pressure drag, C_{D-pre} is the pressure drag coefficient, F_{D-vis} is the viscous drag, C_{D-vis} is the viscous drag coefficient, C_{D-mom} is the jet momentum coefficient in the horizontal (drag) direction, C_{L-mom} is the jet momentum coefficient in the vertical (lift) direction, V_{u-jet} is the velocity of the upper jet, V_{l-jet} is the velocity of the lower jet, respectively.

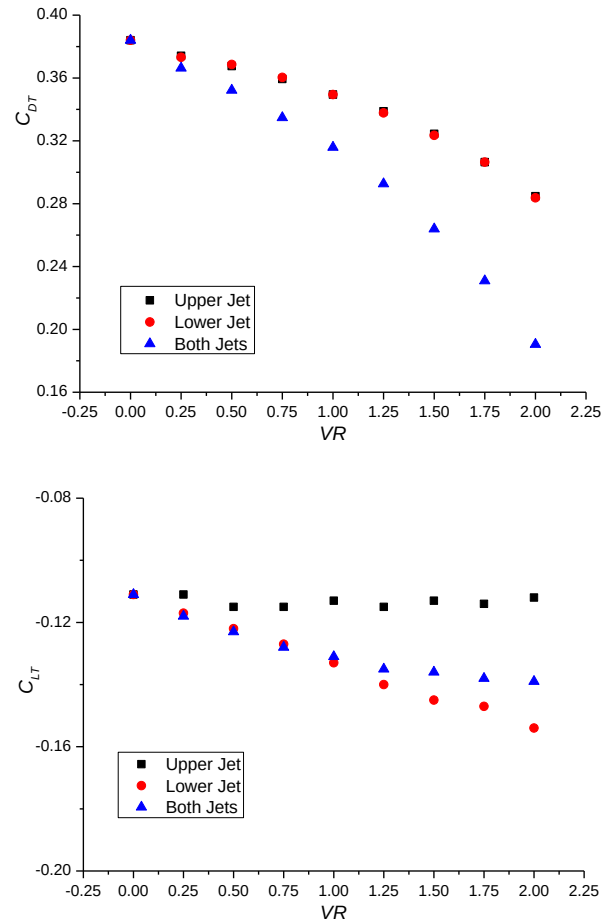


Figure 7. Variations of total drag and lift coefficients as a function of VR for different jet configurations. $\theta=0^\circ$.

Fig. 7 demonstrates the variation of the total drag and lift coefficients with the VR parameter for various jet configurations. In this figure, the inward inclination angle of the jet direction is 0° . The figure indicates that blowing from the rear surfaces has a positive effect in reducing C_{DT} , regardless of the VR and jet configuration. The increase in VR led to a substantial reduction in the C_{DT} . For a value of $VR=2$, the reductions in the C_{DT} are 26%, 25.8%, and 50.5%, respectively, for the lower, upper, and both jet configurations. It has been observed that the combined thrust of both jets is equivalent to the sum of the individual thrusts of each jet.

A close examination of the total lift force graph reveals that C_{LT} is independent of VR in the upper jet configuration. In more precise terms, the jet generated in the upper slot exerts no effect on the lift force within the range from $VR=0$ to 2. In both jet configurations, the total lift exhibited a gradual decrease with increasing VR, up to a value of $VR=1$. For $VR>1$, the rate of this reduction was reduced. It can be posited that the relationship between C_{LT} and VR in this jet configuration is represented by an asymptotic curve. In the lower jet configuration, C_{LT} exhibited a gradual decrease proportional to increasing VR. It has been posited that, in the lower jet configuration, an inverse linear relationship exists between C_{LT} and VR. This finding indicates that the jet produced in the lower slot exerts a substantial influence on the flow emanating from the lower region of the GETS model.

In order to better understand how the resistance and lift coefficients vary with VR, we will first examine how each component contributes to its respective coefficient. Fig. 8 demonstrates the variation of individual drag components with VR. A salient feature of the graphs is that, irrespective of the jet configuration, the viscous component of drag (also known as the friction component) is not a function of the VR parameter. The C_{D-vis} value remains constant at 0.105 in the graphs. It is noteworthy that the variation of the C_{D-pre} component with VR is also quite similar across all three configurations. Irrespective of the jet configuration, the C_{D-pre} decreased and reached a minimum in the VR range from 0.25 to 0.5, after which they increased gradually with increasing VR. Lorite-Díez et al. [20] conducted an investigation into the wake modes of a three-dimensional blunt-based body in proximity to a wall, in the presence of continuous blowing applied at varying locations through four slits along the base edges. The identification of two flow regimes, designated as mass and momentum, was made for all configurations. In the mass flow regime, the reduction in base drag is attributable to small blowing; in the momentum regime, the increase in base drag is attributable to large blowing. A comparison of their base drag coefficient variation-blowing velocity ratio curve with the current pressure drag-velocity ratio curve reveals a high level of similarity. It is also evident that, for $VR>1$, the presence of a rear jet/jets has an opposite effect on the C_{D-pre} (Fig. 8).

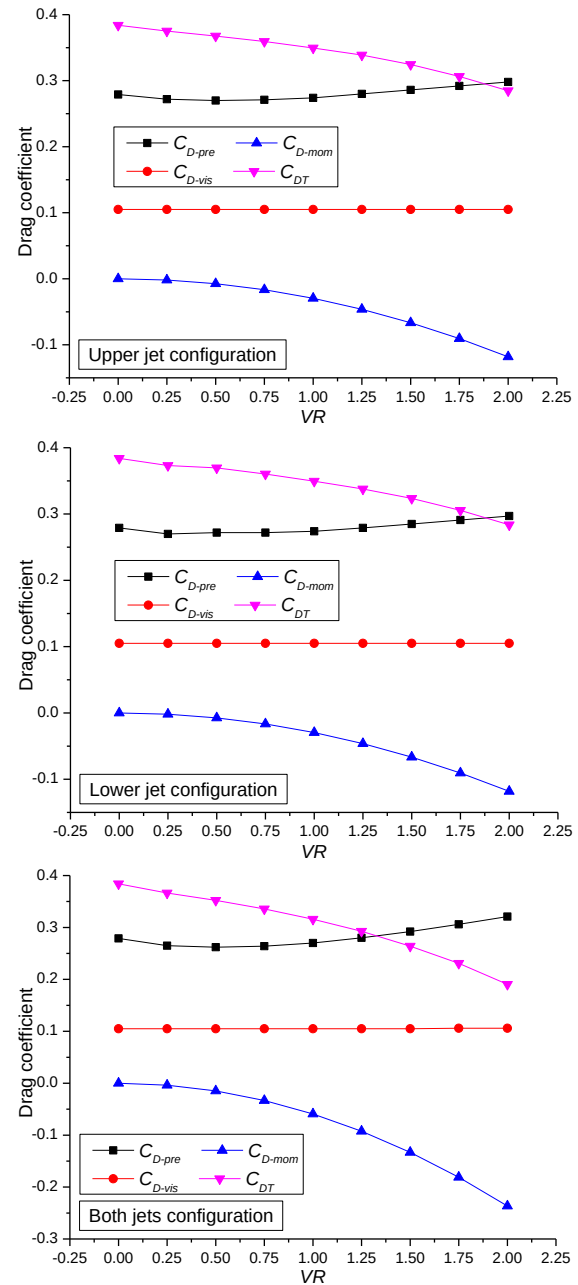


Figure 8. Variation of individual drag components as a function of VR for each jet configuration. $\theta=0^\circ$.

The pressure distribution along the vertical direction on the mid-width of the rear surface is shown in Fig. 9. As can be seen from the figure, the reduction in drag as VR increased to 0.5 is due to an increase in pressure on the rear surface in the range $y/h=0.3-1$. A further increase in VR reduces the base pressure in the y/h range from 0.4 to 1, which increases C_{D-pre} .

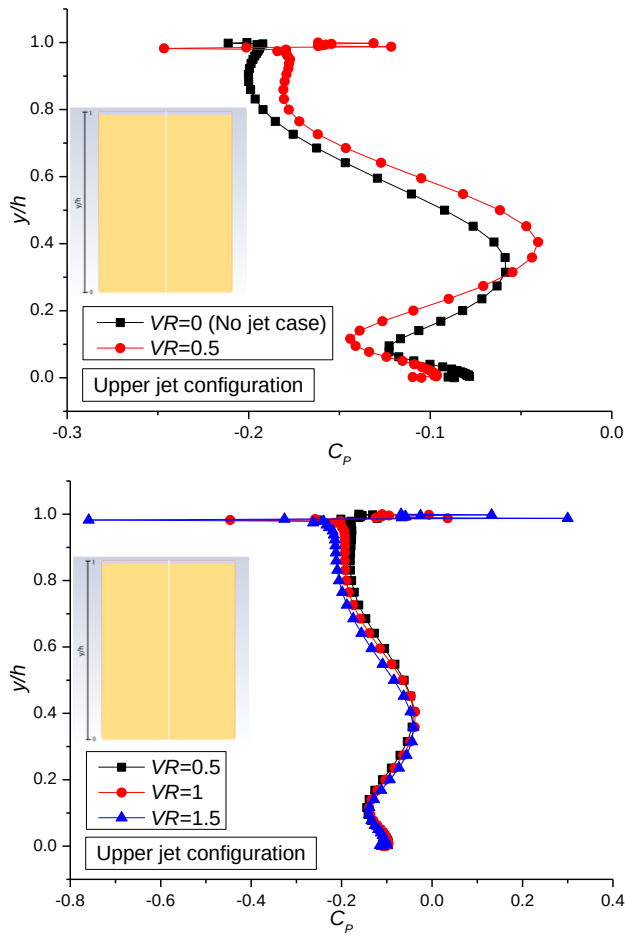


Figure 9. Pressure coefficient distribution along the vertical line on the rear surface of the GETS model. The line is placed on the mid-width of the model, and extends from the bottom trailing edge to the top trailing edge.

Conversely, the jet momentum in the drag direction exerts a substantial influence on the total drag, irrespective of the jet configuration. Irrespective of the VR , this component contributes to the reduction of the total drag caused by the thrust generated by the jet(s). For $VR=2$, the total dimensionless thrust generated by both the upper and lower jets is around -0.24.

As illustrated in Fig. 10, the total lift coefficient and its components vary as a function of VR for each tested jet configuration. In the absence of jet formation behind the GETS model (i.e., $VR=0$), 99.7% of the total lift acting on the GETS model is generated by the C_{L-pre} component. In addition, when $\theta=0^\circ$ and $VR \geq 0$, the jet direction is parallel to the x -axis (drag direction). Consequently, the jet does not generate any force component along the y -axis (lift direction). Consequently, the value of C_{L-mom} is 0. In summary, irrespective of the jet configuration, the C_{L-pre} is the sole contributor to the C_{LT} .

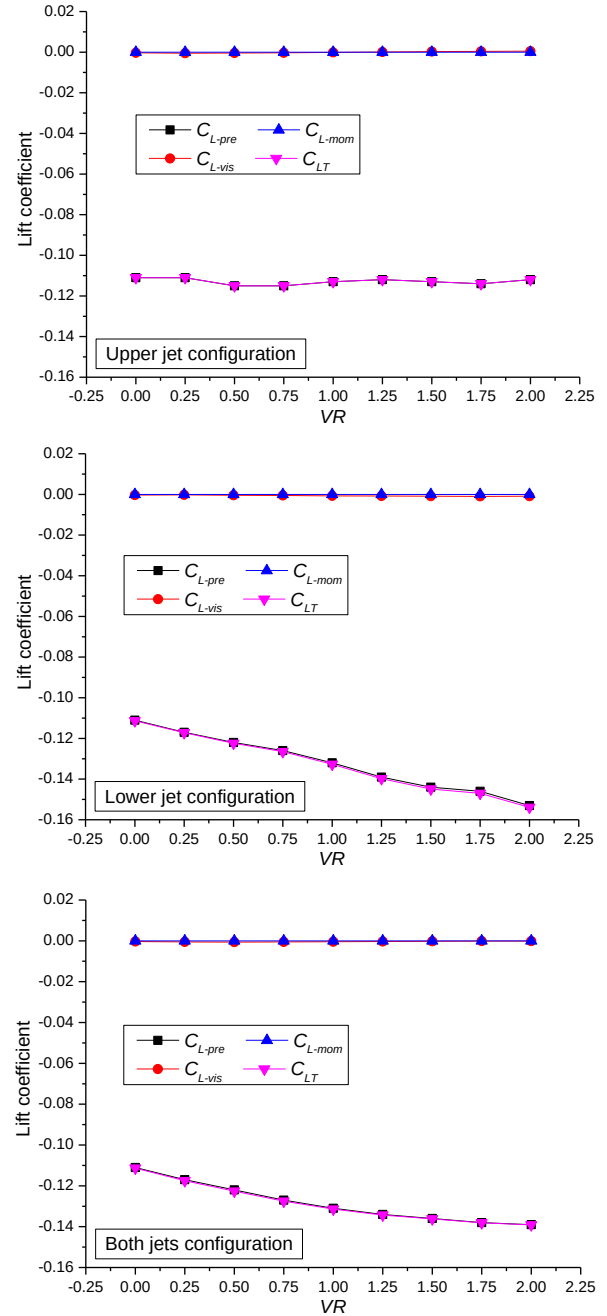


Figure 10. Variation of individual lift components as a function of VR for each jet configuration. $\theta=0^\circ$.

As illustrated in Figs. 8 and 10, C_{LT} remains constant with increasing VR in the upper jet configuration, while it decreases as VR increases in the lower jet and both jet configurations. Fig. 10 demonstrates that the observed change in the C_{LT} is attributable solely to C_{L-pre} . The present study utilizes the lower jet configuration as a foundation to elucidate the underlying cause of the relationship between C_{L-pre} and VR .

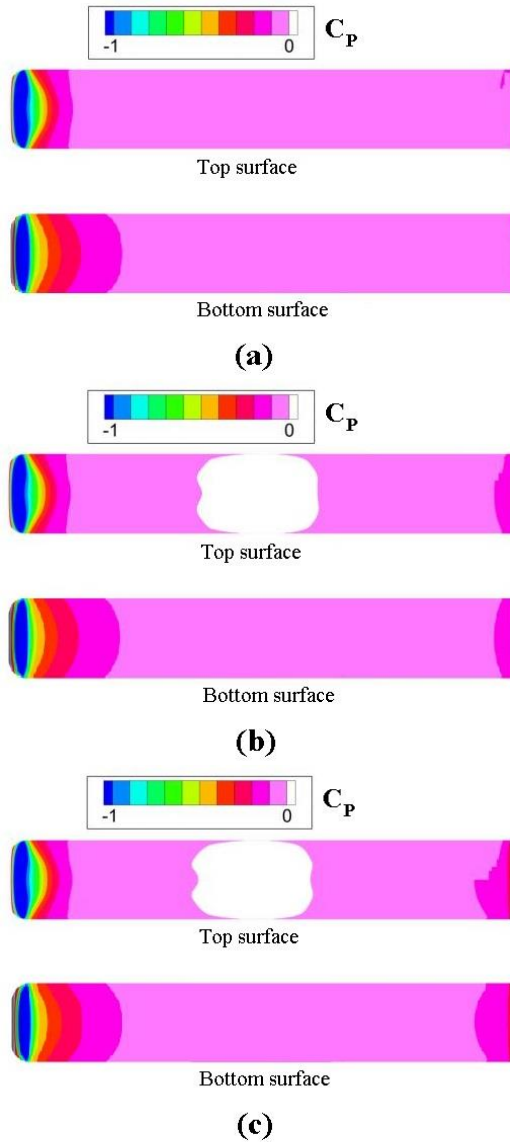


Figure 11. Pressure coefficient distributions on the upper and lower surfaces of the GETS model for the lower jet configuration: (a) $VR=0$ (no jet), (b) $VR=1$, and (c) $VR=2$. $\theta=0^\circ$.

Fig. 11 shows the pressure distributions on the top and bottom surfaces of the GETS model, which contributes most significantly to the C_{L-pre} . Although high pressure exists on the front surface of the model due to the stagnation point where velocity is nearly zero, the flow passes through the rounded portions of the leading edge at high speeds. This high velocity causes low pressure to form in these rounded regions [34]. This explains the low pressure at the front portions of the top and bottom surfaces of the model. Increasing the VR value slightly increases pressure in the middle section of the top surface and decreases the pressure in the rearmost section of the top surface while significantly decreasing pressure in the rearmost section of the lower surface (i.e., near the lower jet location).

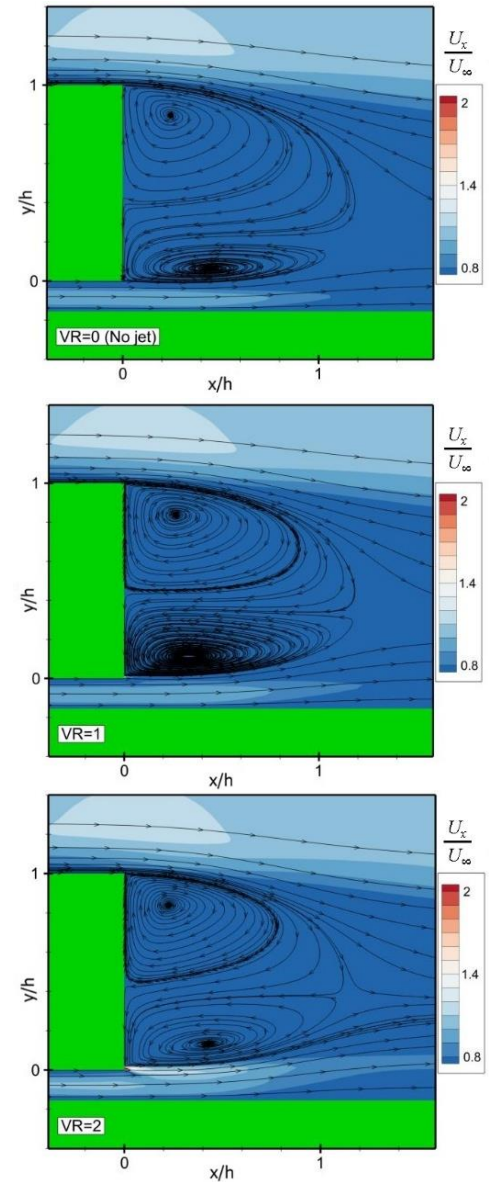


Figure 12. Dimensionless streamwise velocity contours (U_x/U_∞) and streamline topology on the x - y plane, which divides the GETS model into two equal parts for the lower jet configuration. $\theta=0^\circ$.

Fig. 12 shows the dimensionless streamwise velocity contours and streamline topology in the rear section of the GETS model for the lower jet configuration. These contours and topologies were obtained from the x - y plane at $z=0$ and illustrate the effects of the lower jet, with different VR values, on the flow structure. The first noticeable observation in the figure is that the sizes of the large recirculation cells change as VR increases. Another notable observation is that an increase in VR causes the deceleration distance of the flow exiting the lower part of the model to lengthen. Furthermore, and most importantly, an increase in VR causes the flow at the bottom of the model to approach the bottom surface before it has even emerged from the rear section, thereby causing a drop in pressure in this region.

We will now examine how the inclination angle of the jet direction, which varies between -16° and $+16^\circ$ for two VR values (i.e., 0.5 and 1), affects the aerodynamic characteristics. The effects of θ on total drag are presented in Fig. 13 for various jet configurations. Regardless of the tested θ , it is evident from the figure that the drag acting on the GETS model with any jet configuration is lower than that of the GETS model with no jet (i.e., $VR=0$). Regardless of the VR and jet configuration, the gradual and linear increase in the total drag with the decrease in θ from 0 to -16° is evident in the figure. Conversely, increasing θ in the positive range (i.e., from 0 to 16°) generally leads to a slight decrease in total drag. Consequently, increasing θ in the positive range (i.e., directing the jet toward the base of the model) generally has a positive effect on drag reduction.

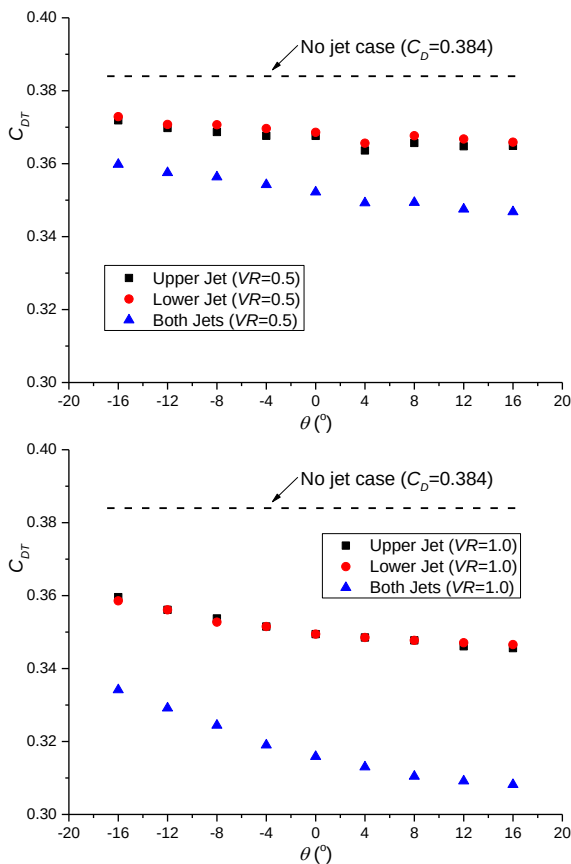


Figure 13. Variations of total drag coefficients as a function of θ for different jet configurations. $VR=0.5$ and 1.

The subsequent discussion will concentrate on the manner in which each component contributes to the total drag values obtained as a function of θ for these different VR values. In order to address this objective, the upper and both jet configurations were considered (Figs. 14 and 15). It is evident from Figs. 14 and 15 that the viscous drag exerted on the model is independent of θ , irrespective of the VR and jet configuration. It is evident that C_{D-mom} exerts a beneficial impact on drag reduction, irrespective of the θ tested. The x-component of the jet momentum

undergoes a slight alteration as the θ increases in a positive or negative direction. This outcome is not unexpected, given that only the lower values of θ were investigated. As the θ value increased from -16 to $+16^\circ$, the C_{D-pre} reduction occurred in a proportionate yet gradual manner. It is evident that as the value of VR rises from 0.5 to 1, the rate of reduction in the C_{D-pre} also increases. The findings indicate that both C_{D-mom} and C_{D-pre} are responsible for the reduction in the C_{DT} . Furthermore, the contribution of C_{D-pre} to the reduction in C_{DT} attains its maximum level at approximately $\theta=+16^\circ$, while that of C_{D-mom} reaches its maximum level at approximately $\theta=0^\circ$.

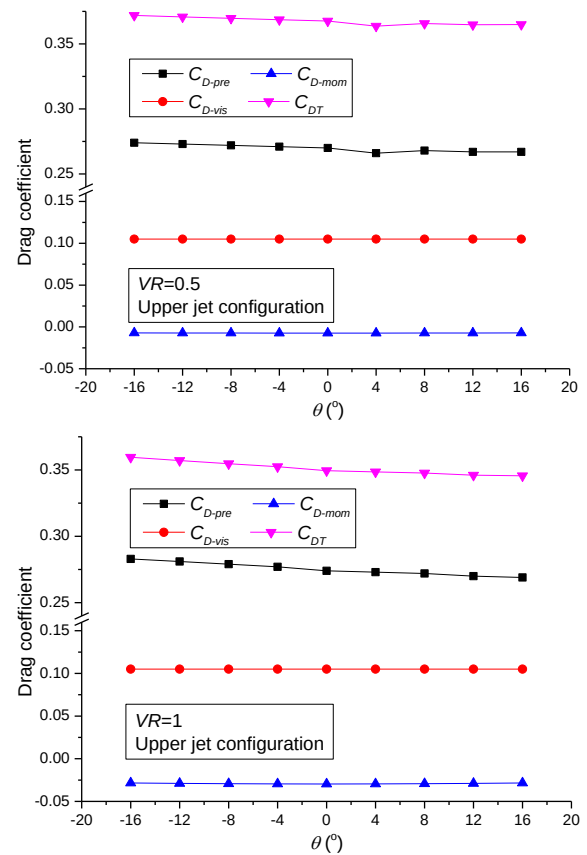


Figure 14. Variation of individual drag components as a function of θ for the upper jet configuration. $VR=0.5$ and 1.

Fig. 16 illustrates the pressure distribution on the rear surface of the GETS model for upper jet and both jets configurations. For the upper jet case, an increase in θ from -16° to $+16^\circ$ results in the following outcomes. The relatively high-pressure region in the mid-height of the rear surface undergoes expansion. The pressure of the region near the lower edge of the rear surface increases. The two regions coalesce at the highest θ . It is noteworthy that the increase in θ (i.e., the inward inclination of the upper jet) results in a concomitant increase in pressure in the lower half of the rear surface. In the lower jet case, the relatively high-pressure region located just below the half-height of the rear surface shifts toward the center of the rear surface as θ is

increased (not shown here) and expands. Furthermore, an increase in the pressure of the upper half part near the jet and a decrease in the pressure of the lower half part near the jet were observed. In contrast to the upper jet configuration, the lower jet configuration exhibited a symmetrical distribution of pressures in the upper and lower halves of the model's rear section when the θ increased.

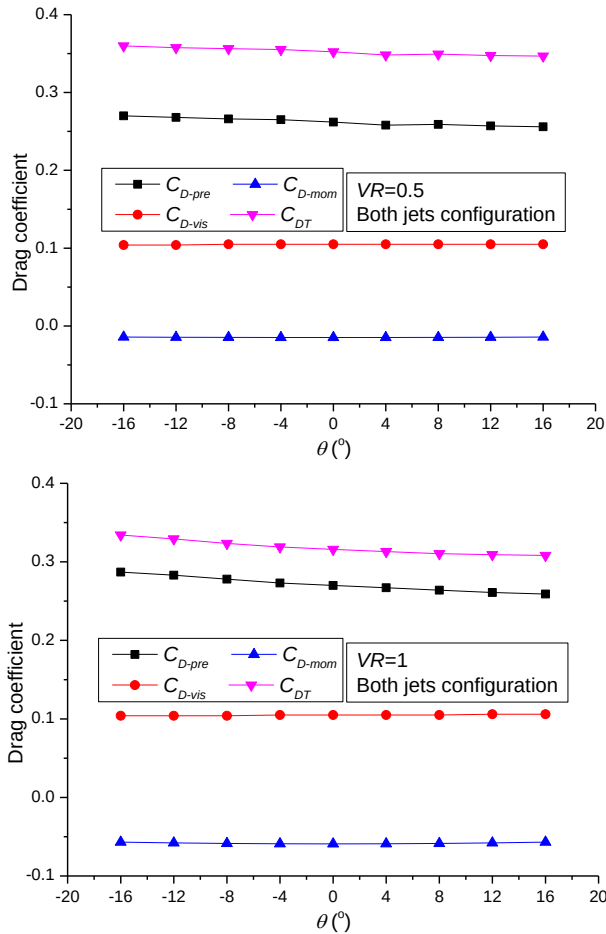


Figure 15. Variation of individual drag components as a function of θ for the both jets configuration. $VR=0.5$ and 1.

In the both jets configuration, an increase in θ results in a shift of the relatively high-pressure region, located just below the central part of the rear surface, towards the center of the rear surface, accompanied by an expansion of the region. The increase in pressure is discernible in both the lower and upper regions of the rear surface (Fig. 17). Irrespective of the jet configuration, an increase in the area-weighted pressure at the rear surface is observed with an increase in θ . This increase serves as the underlying cause of the decrease in the C_{D-pre} with increasing θ .

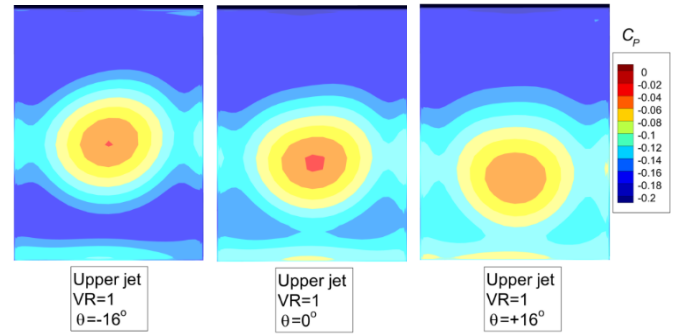


Figure 16. Pressure contours on the rear (base) surface of the GETS model for various θ . Upper jet configuration and $VR=1$.

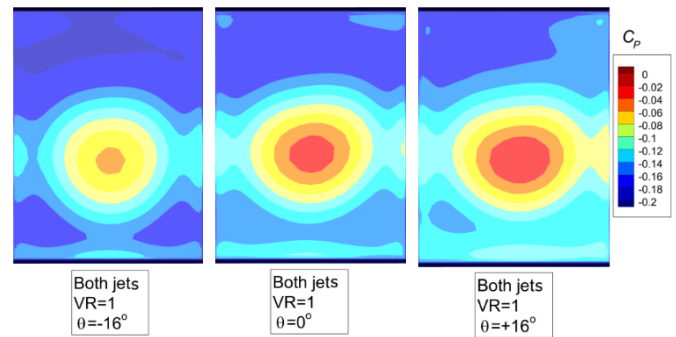


Figure 17. Pressure contours on the rear (base) surface of the GETS model for various θ . Both jets configuration and $VR=1$.

The effects of θ on total lift are presented in Fig. 18 for various jet configurations and VR . It is noteworthy that the C_{LT} exhibits a linear relationship with theta, irrespective of the jet configuration. An increase in the VR value from 0.5 to 1 results in a more precipitous line representing the VR - θ relationship, as illustrated in the figure. In the context of lower jet and both jets configurations, an augmentation in θ has been observed to result in a concomitant reduction in the C_{LT} . This decline in C_{LT} has been shown to enhance the traction of vehicle tires, while concurrently leading to an increase in fuel consumption. In the upper jet configuration, an increase in the C_{LT} with θ has been shown to improve fuel efficiency but reduce tire traction.

Fig. 19 shows the contribution of each component to the total lift values obtained as a function of θ , at $VR=1$. In order to address this objective, all three jet configurations were considered. As demonstrated in the figure, it is evident that there is no observable impact of C_{L-vis} on C_{LT} . The C_{L-mom} component exerts a negligible influence on the C_{LT} for lower and upper jet configurations. This limitation is not unexpected, given that the maximum jet inclination angle is $\pm 16^\circ$, which is considerably less than $\pm 90^\circ$. In the both jets configuration, the C_{L-mom} generated by the upper and lower jets act in opposite directions, resulting in the C_{L-mom} not contributing to the C_{LT} in this configuration.

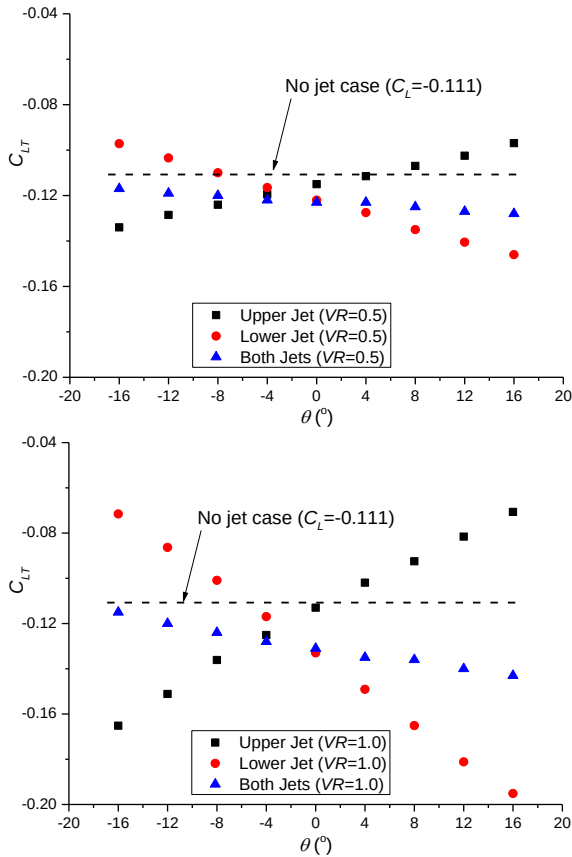


Figure 18. Variations of total lift coefficients as a function of θ for different jet configurations. VR=0.5 and 1.

As illustrated in Fig. 19, the C_{L-pre} is the primary contributor to the C_{LT} . This finding indicates that the pressure variations between the upper and lower surfaces of the GETS model are crucial for the generation of lift, particularly in the upper and lower jet configurations. The aforementioned pressure differences for θ equals -16 and +16° are displayed in Fig. 20. The pressure coefficient (C_p) values depicted in the graphs of the figure are derived from the mid-width of the top and bottom surfaces of the model (see yellow line in the figure). The distribution of C_p on the top and bottom surfaces is consistent across the x-axis, ranging from -0.615 m (aft point) to approximately -0.150 m, irrespective of the theta and jet configuration. Within this range, a negative lift, also known as downforce, is generated. It can be posited that the variation in C_p distributions within the remaining range of -0.150 m to 0 m (rear edges) exerts a more substantial influence on the determination of the C_{L-pre} as well as the C_{LT} . In the upper jet case, the flow on the top surface was partially obstructed by the jet at $\theta = -16^\circ$. This results in an increase in pressure in the area adjacent to the rear edge of the top surface. In this case, the pressure distribution in the bottom surface of the model exhibits the same distribution along the x-coordinate, regardless of θ . At $\theta = +16^\circ$, a vacuum pressure was generated in the vicinity of the rear edge of the top surface. This phenomenon results in a lift generation, thereby elucidating the reduction in downforce observed as θ increases from -16° to $+16^\circ$.

In the lower jet case, the pressure distribution is reverse to that observed in the upper jet case (Fig. 20). In this case, the distribution of pressure in the top section of the model remains constant, irrespective of θ .

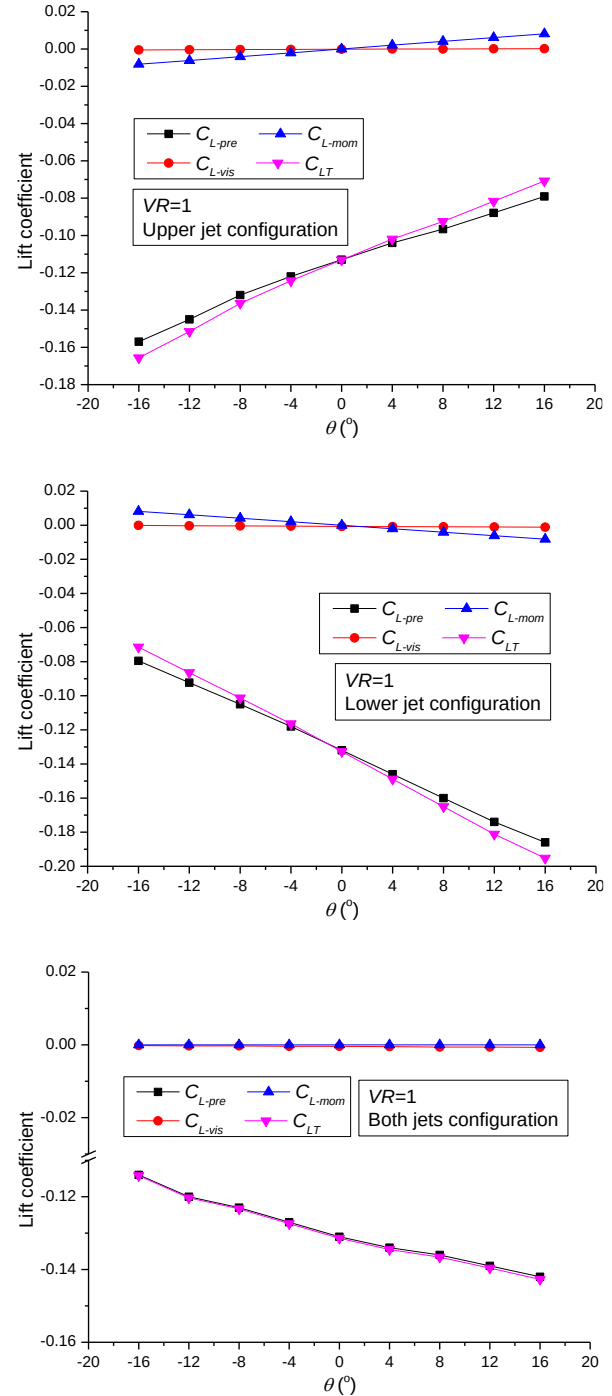


Figure 19. Variation of individual lift components as a function of θ for all jet configurations tested. VR=1.

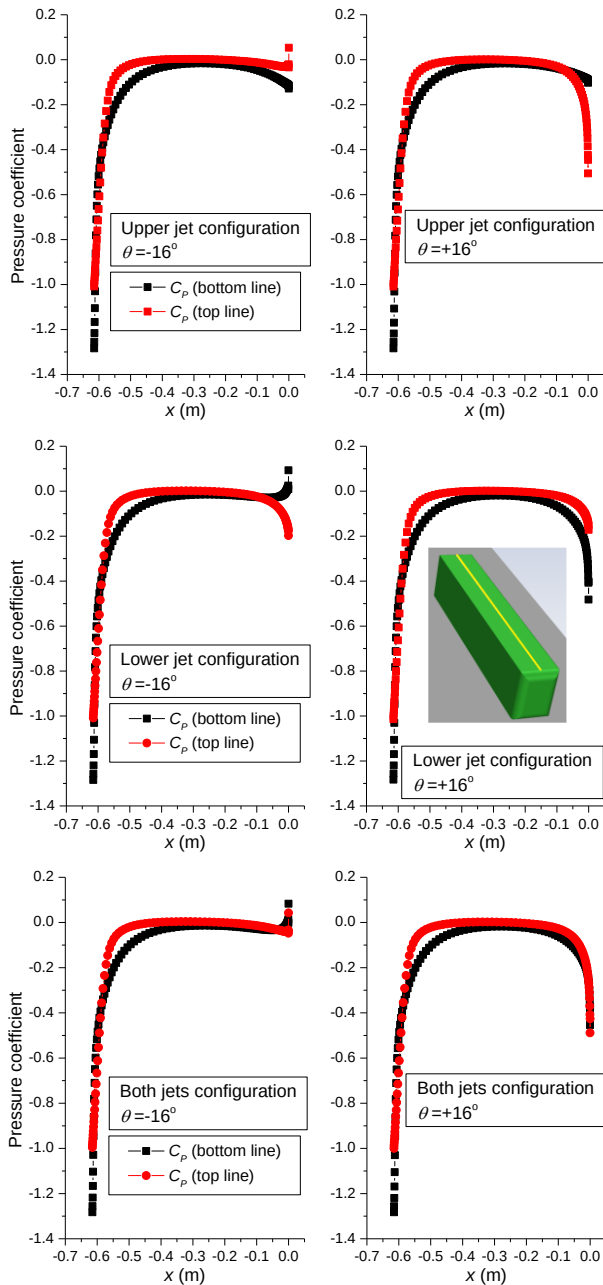


Figure 20. Pressure distribution on the mid-length of the top and bottom surfaces of the GETS model for various jet configurations tested. $\theta = -16^\circ$ and $+16^\circ$. $VR=1$.

Conversely, the downward flow of the jet stream at $\theta = -16^\circ$ obstructed the flow passing through the lower part of the model, which in turn induced a pressure increase in the rearmost section of the model's bottom. However, at an angle of $+16^\circ$, the jet has transferred momentum to the flow emanating from beneath the model, thereby causing an acceleration in the model and consequently leading to a pressure drop in the lower rear section. This elucidates the underlying reason for the enhancement in downforce with an augmentation in θ . In the both jets configuration, there is a slight increase in downforce due to the lower part of

the model being in close proximity to the ground. The absence of a substantial rise in the lift force exerted on the model as θ increases in the both jets configuration can be attributed to the generation of opposing forces along the vertical axis by the upper and lower surfaces.

3.2. Active control efficiency

It is evident that the aerodynamic drag of the simplified HDV model can be decreased by blowing from the rear slits. The provision of this blowing (i.e., the generation of these jets on the rear surface of the model) necessitates an additional energy expenditure. The efficacy of active flow control can be substantiated by a comparison of the energy recovered from this reduction in aerodynamic drag with the energy required to generate the jet. The following formula, based on the blowing system presented in Fig. 21, has been demonstrated to provide this efficiency.

$$\eta_{control} = \frac{\Delta F_D \cdot U_\infty}{\Delta P \cdot Q / \eta_{comp}} = \frac{\Delta F_D \cdot U_\infty}{0.5 \cdot \rho_2 \cdot V_{jet}^3 \cdot A_s / \eta_{comp}} \quad (9)$$

where ΔF_D is the absolute aerodynamic drag difference between the model with and without the control, ΔP is the pressure difference between the points given in the figure, Q is the flow rate, ρ_2 is the density of air at the slot exit (point 2), and η_{comp} is the compressor efficiency ($\eta_{comp}=0.70$ in this study). The expression for the power necessary to generate a jet at the desired velocity in the slot was derived using the Bernoulli equation and under the assumption that changes in air density can be disregarded.

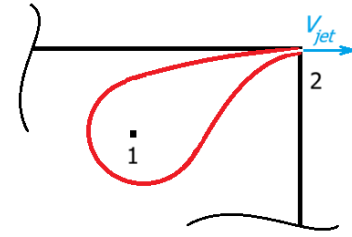


Figure 21. Side view of the upper rear part of the GETS model. The system used to generate jet is depicted.

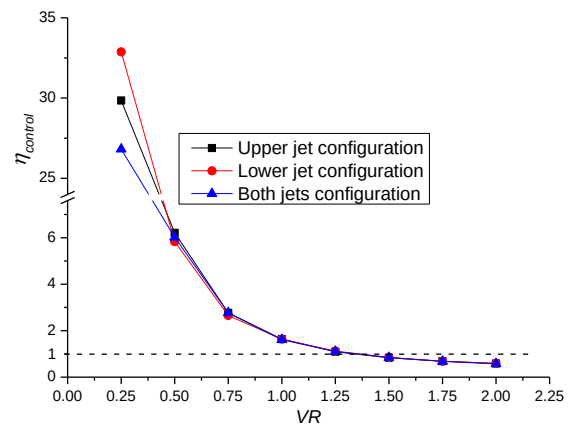


Figure 22. Active flow control efficiencies of various jet configurations as a function of VR . $\theta=0^\circ$ and $\eta_{comp}=0.70$.

Eq. (9) demonstrates that the power necessary to generate a jet in the rear slot is directly proportional to the cube of the jet exit velocity (V_{jet}). This phenomenon elucidates the underlying reason for the observed decline in control efficiency ($\eta_{control}$) with increasing VR. It has been demonstrated that control efficiency is not contingent upon jet configuration for $VR \geq 0.5$. For values of VR less than around 1.25, the control efficiency exceeds 1. However, for VR values greater than 1.5, the control efficiency is less than 1. In essence, the employment of the active flow control method for $VR \leq 1.25$ is advantageous in terms of efficiency.

4. Conclusions

In this study, a simplified tractor-trailer combination model was modified by creating narrow slits along its rear upper and lower edges, where planar jets were generated. The effects of the positions, velocities, and exit directions of these jets on the aerodynamic characteristics of the aforementioned model were investigated using steady-flow simulations. The following results were obtained from this study.

When the jet flow direction is aligned with the free-stream direction ($\theta=0^\circ$), employing the upper jet facilitates a reduction in the drag acting on the model without inducing any change in the value of the lift force acting on the model.

The lower jet interacts with the flow between the model's bottom surface and the ground. As θ varies for a fixed VR value, it causes a significant and inevitable change in the lift force acting on the model.

In the upper jet configuration, at a fixed value of VR, an increase in θ in the positive direction results in an increase in lift, while an increase in θ in the negative direction results in a decrease in lift. In the lower jet configuration, the situation is precisely the reverse.

The viscous components of the total aerodynamic characteristics are independent of the jet configuration, VR, and θ .

The both jet configuration, which utilizes both the upper and lower jets concurrently, yielded the lowest aerodynamic drag values for the model.

When the θ value is set to 0° , the application of the active flow control method while maintaining a VR value below 1.25 is essential to ensure the effective implementation of the method.

It is also imperative to address a salient aspect associated with the findings of this study in this section. As previously mentioned, the Reynolds number independence criterion has not been met in the present numerical study. The findings of the present numerical study demonstrate that the utilization of both jet configurations with $VR=2$ and $\theta=0^\circ$ results in a 50% reduction in aerodynamic drag. In real-world applications, the probability of achieving such a substantial reduction in aerodynamic drag without implementing any radical design modifications is rather low. However, the current numerical study indicates that a reduction in aerodynamic drag, albeit much lower than 50%, can be achieved.

Nomenclature

A_F	: frontal area of the GETS model (m^2)
A_s	: area of the slot (m^2)
CC	: specific limiter for the curvature correction function (-)
C_D	: drag coefficient (-)
C_{D-mom}	: jet momentum coefficient in the horizontal (drag) direction (-)
C_{D-pre}	: pressure drag coefficient (-)
C_{D-vis}	: viscous drag coefficient (-)
C_{DT}	: total drag coefficient (-)
C_L	: lift coefficient (-)
C_{L-mom}	: jet momentum coefficient in the vertical (lift) direction (-)
C_{L-pre}	: pressure lift coefficient (-)
C_{L-vis}	: viscous lift coefficient (-)
C_{LT}	: total lift coefficient (-)
C_p	: pressure coefficient (-)
C_μ	: momentum coefficient (-)
F_D	: drag (N)
F_{D-pre}	: pressure drag (N)
F_{D-vis}	: viscous drag (N)
F_L	: lift (N)
F_{L-pre}	: pressure lift (N)
F_{L-vis}	: viscous lift (N)
g	: clearance between the GETS model and the ground plate (m)
h	: height of the GETS model (m)
l	: length of the GETS model (m)
Re_w	: width-based Reynolds number (-)
Tu_x	: freestream turbulence intensity (-)
U_∞	: freestream velocity (m/s)
U_x	: streamwise velocity component (m/s)
V_{jet}	: jet velocity (m/s)
V_{l-jet}	: velocity of the lower jet (m/s)
V_{u-jet}	: velocity of the upper jet (m/s)
VR	: velocity ratio (-)
w	: width of the GETS model (m)
x	: streamwise coordinate axis (m)
y	: transverse coordinate axis (m)
y^+	: non-dimensionalized distance from the wall (-)
z	: spanwise coordinate axis (m)
$\eta_{control}$	: active flow control efficiency
η_{comp}	: compressor efficiency
θ	: inward inclination angle of the jet direction with respect to the main flow ($^\circ$)
μ	: viscosity of air ($Pa \cdot s$)
ρ	: density of air (kg/m^3)
ρ_2	: density of air at the slot exit (kg/m^3)
CFD	: computational fluid dynamics
EU	: european union
GETS	: generalized european transport system
GHG	: greenhouse gas
HDV	: heavy-duty vehicle
NACA	: national advisory committee for aeronautics
RANS	: reynolds-averaged navier-stokes
SaOB	: synchronized suction and oscillatory blowing

SIMPLEC : semi-implicit method for pressure linked equations-consistent
SST : shear stress transport
QUICK : quadratic upstream interpolation for convective kinematics

Conflict of Interest Statement

The authors declare that there is no conflict of interest in the study.

CRediT Author Statement

Erhan Firat: Conceptualization, Data curation, Formal analysis, Investigation, Software, Validation, Writing-original draft, Writing - review & editing

Mehmet Seyhan: Resources, Formal analysis, Investigation, Validation, Writing-original draft, Writing - review & editing

Declaration of Generative AI in the Writing Process

During the preparation of this manuscript, the authors used DeepL Write for the purpose of enhancing the quality of the writing. All AI-assisted output was subsequently reviewed and, where necessary, edited by the authors, who take full responsibility for the final content of the manuscript.

References

- [1] European Union. Road transport. Climate Action. 2023.
- [2] Hariram A, Koch T, Mardberg B, Kyncl J. A Study in Options to Improve Aerodynamic Profile of Heavy-Duty Vehicles in Europe. Sustainability. 2019;11(19):5519. <https://doi.org/10.3390/su11195519>
- [3] McAuliffe BR. Improving the aerodynamic efficiency of heavy-duty vehicles: wind tunnel test results of trailer-based drag-reduction technologies. National Research Council of Canada, Aerospace, Aerodynamics Laboratory; 2015 Jul. p. 95. <https://doi.org/10.4224/21275397>
- [4] Browand F, Radovich C, Boivin M. Fuel Savings by Means of Flaps Attached to the Base of a Trailer: Field Test Results. SAE Technical Paper 2005. <https://doi.org/10.4271/2005-01-1016>
- [5] Salati L, Schito P, Cheli F. Wind tunnel experiment on a heavy truck equipped with front-rear trailer device. Journal of Wind Engineering and Industrial Aerodynamics. 2017;171:101-109. <https://doi.org/10.1016/j.jweia.2017.09.016>
- [6] Firat E. Aerodynamic Drag Improvements on a Simplified Heavy Vehicle using Three-sided Plain and Notched Base Flaps. Journal of Applied Fluid Mechanics. 2023;16(7):1467-1482. <https://doi.org/10.47176/jafm.16.07.1748>
- [7] Landman D, Wood R, Seay W, Bledsoe J. Understanding Practical Limits to Heavy Truck Drag Reduction. SAE Int. J. Commer. Veh. 2010;2(2):183-190. <https://doi.org/10.4271/2009-01-2890>
- [8] Bukreev KR. Shape optimisation of teardrop trailers to minimise aerodynamic drag in articulated lorries. International Journal of Thermofluids. 2023;18:100334. <https://doi.org/10.1016/j.ijft.2023.100334>
- [9] Phan TL, Nguyen TT, Nguyen THT. Optimization of unsteady jet control flow method for aerodynamic drag reduction of heavy truck model. Results in Engineering. 2024;22:102167. <https://doi.org/10.1016/j.rineng.2024.102167>
- [10] Vernet JA, Örlü R, Söderblom D, Elofsson P, Alfredsson PH. Plasma Streamwise Vortex Generators for Flow Separation Control on Trucks. Flow, Turbulence and Combustion. 2018;100:1101-1109. <https://doi.org/10.1007/s10494-018-9891-9>
- [11] Minelli G, Hartono EA, Chernoray V, Hjelm L, Krajnović S. Aerodynamic flow control for a generic truck cabin using synthetic jets. Journal of Wind Engineering and Industrial Aerodynamics. 2017;168:81-90. <https://doi.org/10.1016/j.jweia.2017.05.006>
- [12] Seifert A, Shtendel T, Dolgopyat D. From lab to full scale Active Flow Control drag reduction: How to bridge the gap? Journal of Wind Engineering and Industrial Aerodynamics. 2015;147:262-272. <https://doi.org/10.1016/j.jweia.2015.09.012>
- [13] Bardera R, Matías-García JC, Barroso-Barderas E, Fernández-Antón J, Rodríguez-Sevillano AA. Active Flow Control by Coanda Actuators for Aerodynamic Drag Reduction in a European-Type Truck. Actuators. 2025;14(11):556. <https://doi.org/10.3390/act14110556>
- [14] Manosalvas DE, Economon TD, Othmer C, Jameson A. Computational Design of Drag Diminishing Active Flow Control Systems for Heavy Vehicles. In: 8th AIAA Flow Control Conference. 2016. p. 1-15. <https://doi.org/10.2514/6.2016-4082>
- [15] Littlewood RP, Passmore MA. Aerodynamic drag reduction of a simplified squareback vehicle using steady blowing. Experimental Fluids. 2012;53(2):519-529. <https://doi.org/10.1007/s00348-012-1306-4>
- [16] Aubrun S, McNally J, Alvi F, Kourta A. Separation flow control on a generic ground vehicle using steady microjet arrays. Experimental Fluids. 2011;51(5):1177-1187. <https://doi.org/10.1007/s00348-011-1132-0>
- [17] Baek SW, Lee SW. Aerodynamic drag reduction on a realistic vehicle using continuous blowing. Microsystem Technologies. 2019;26:11-23. <https://doi.org/10.1007/s00542-019-04355-w>
- [18] Liu S, Chen T, Zhou W. Numerical Analysis of Aerodynamic Drag Reduction for a DrivAer Automobile Model Using Rear Air Jets. Applied Sciences. 2025;15(22):12334. <https://doi.org/10.3390/app152212334>
- [19] Cerutti JJ, Sardu C, Cafiero G, Iuso G. Active Flow Control on a Square-Back Road Vehicle. Fluids. 2020;5(2):55. <https://doi.org/10.3390/fluids5020055>
- [20] Lorite-Díez M, Jiménez-González JJ, Pastur L, Martínez-Bazán C, Cadot O. Experimental analysis of the effect of local base blowing on three-dimensional wake modes. Journal of Fluid Mechanics. 2020;883:A53. <https://doi.org/10.1017/jfm.2019.917>

- [21] McNally JP, Fernandez EJ, Robertson GJ, Kumar R, Taira K, Alvi FS, et al. Drag reduction on a flat-back ground vehicle with active flow control. *Journal of Wind Engineering and Industrial Aerodynamics*. 2015;145:292-303. <https://doi.org/10.1016/j.jweia.2015.03.006>
- [22] Corallo M, Sheridan J, Thompson MC. Effect of aspect ratio on the near-wake flow structure of an Ahmed body. *Journal of Wind Engineering and Industrial Aerodynamics*. 2015;147:95-103. <https://doi.org/10.1016/j.jweia.2015.09.006>
- [23] Zhang J, Wang F, Guo Z, Han S, Gao G, Wang J. Investigation of the wake flow of a simplified heavy vehicle with different aspect ratios. *Physics of Fluids*. 2022;34:065135. <https://doi.org/10.1063/5.0094534>
- [24] Hassaan M, Badlani D, Nazarinia M. Numerical study of the effect of aspect ratio on the flow characteristics of the Ground Transportation System. *Journal of Wind Engineering and Industrial Aerodynamics*. 2020;206:104314. <https://doi.org/10.1016/j.jweia.2020.104314>
- [25] Van Raemdonck GMR, Van Tooren MJL. Time-Averaged Phenomenological Investigation of a Wake Behind a Bluff Body. In: *Sixth International Colloquium on Bluff Body Aerodynamics & Applications*. Milano, Italy; 2008.
- [26] SAE International. SAE Wind Tunnel Test Procedure for Trucks and Buses. SAE Standard J1252_201207. Revised July 2012. https://doi.org/10.4271/J1252_201207
- [27] Hassaan M, Badlani D, Nazarinia M. On the effect of boat-tails on a simplified heavy vehicle geometry under crosswinds. *Journal of Wind Engineering and Industrial Aerodynamics*. 2018;183:172-186. <https://doi.org/10.1016/j.jweia.2018.10.013>
- [28] Mosiężny J, Ziegler B, Grzymisławski P, Ślęfarski R. Base drag reduction concept for commercial road vehicles. *Energy*. 2020;205:118075. <https://doi.org/10.1016/j.energy.2020.118075>
- [29] İpci D. A Comparative CFD Study of Side-view Mirror and Side-view Camera Usages on a City Bus. *International Journal of Automotive Science and Technology*. 2020;4(3):138-143. <https://doi.org/10.30939/ijastech..726376>
- [30] Çalışkan M, Bakırcı A. An Investigation of The Different Diffuser Positions Effect on Vehicle Aerodynamic Performance. *International Journal of Automotive Science and Technology*. 2022;6(1):75-82. <https://doi.org/10.30939/ijastech..1073376>
- [31] Dağdeviren A, Gürsoy E, Gedik E, Arslan K. Heat transfer and entropy analysis of Fe₃O₄/H₂O ferrofluid in tear-drop-dimpled circular tubes. *Journal of Thermal Analysis and Calorimetry*. 2025;150:19657-19676. <https://doi.org/10.1007/s10973-025-14972-5>
- [32] Menter FR. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA Journal*. 1994;32(8):1598-1605. <https://doi.org/10.2514/3.12149>
- [33] Van Straaten M. Computational and Experimental Investigation on Base Drag Reduction of a Generic Transportation System by Guiding Vanes [thesis]. Delft University of Technology; 2007.
- [34] Van Tilborg F, Van Raemdonck G, Sciacchitano A, Casalino D. Flow Analysis between Two Bluff Bodies in a Close Distance Platooning Configuration. *SAE International Journal of Commercial Vehicles*. 2019;12(3):179-196. <https://doi.org/10.4271/02-12-03-0015>