

Numerical study of the influence of duct cross-section geometry on flow characteristics and brake cooling performance in a heavy-duty bus

Edinson Bryan Velasquez Quispe¹, Ahsly Britney Wagner Laguna¹,
Juan Carlos Valdez Loaiza¹ , José Canazas^{1*} 

¹ Universidad Nacional de San Agustín de Arequipa, Arequipa 04000, Perú

* Corresponding author's e-mail: jcanazas@unsa.edu.pe

ABSTRACT

Efficient heat dissipation during braking is crucial for maintaining the performance and reliability of brake systems in heavy-duty vehicles. Although CFD has been widely applied to brake thermal analysis, the isolated influence of brake cooling duct cross-sectional geometry on airflow characteristics and convective cooling performance has received limited attention. This study numerically evaluated the influence of duct cross-sectional geometry on the flow characteristics and cooling performance of a heavy-duty bus brake system using two consecutive computational fluid dynamics (CFD) models implemented in ANSYS CFX. First, circular and square cross-section ducts were analyzed for vehicle speeds of 60, 90, and 120 km/h to determine the outlet flow conditions. Subsequently, these flow conditions were used as inlet boundary conditions for the thermal analysis of a brake disc subjected to surface temperatures of 600, 700, and 800 K. The numerical reliability of the simulations was verified through mesh independence analyses, where successive mesh refinements produced variations below 0.5% in the monitored variables. The results showed that the circular duct generated higher exit velocities than the square duct under all analyzed conditions, reaching 51.88 m/s and 39.52 m/s, respectively, at 90 km/h. Although this improvement was accompanied by greater pressure losses, the circular configuration increased the brake disc heat dissipation by approximately 51% at a surface temperature of 600 K. These findings demonstrate that duct geometry plays a decisive role in brake cooling performance and provide useful engineering guidelines for designing more efficient brake cooling systems for heavy-duty buses operating under severe thermal conditions.

Keywords: brake cooling, heavy-duty bus, cooling duct geometry, computational fluid dynamics, convective heat transfer.

INTRODUCTION

The braking system is a critical component in road transport, ensuring vehicle safety by converting kinetic energy into thermal energy through friction between the brake pads and the disc [1–3]. During repeated or high-intensity braking, a significant amount of heat is generated at the braking interface, leading to a temperature increase that can compromise braking effectiveness, accelerate material wear, and, in extreme cases, cause brake failure [4–6]. Furthermore, the heat generated during severe braking may increase the

brake fluid temperature, adversely affecting its hydraulic performance and increasing the risk of brake fade or vapor lock [7,8]. Therefore, effective thermal management of brake discs is essential to maintain performance and reliability under various operating conditions.

To mitigate excessive temperature increases, several cooling strategies have been developed, including ventilated brake discs, improved material designs, and the use of forced convection via airflow [9,10]. Among these, the implementation of air ducts to direct external airflow toward the brake assembly has become established as a

practical and cost-effective solution, especially in high-performance and heavy-duty vehicles [11,12]. However, the effectiveness of these systems depends heavily on the duct's geometric configuration, which governs the airflow distribution, velocity profiles, and ultimately, convective heat transfer across the disc surface [13,14].

In recent years, numerical methods based on computational fluid dynamics (CFD) have been widely used to investigate the thermofluid behavior of braking systems. Several studies have analyzed heat dissipation and convective heat transfer in brake discs under transient operating conditions, highlighting the ability of coupled models to predict the thermal evolution of the system during braking cycles, as presented in Chaudhary et al. [15]. Likewise, recent research has addressed the aerothermal analysis of the braking system considering complete vehicle geometries and realistic flow conditions, demonstrating the significant influence of external aerodynamics on heat dissipation mechanisms [16]. Other studies have delved deeper into the evaluation of heat transfer and flow patterns using CFD simulations applied to specific thermal configurations, providing relevant information on convective coefficients and temperature distribution in air-cooled systems [17]. Other studies have incorporated multi-physics approaches that consider fluid-structure interaction and dynamic effects during braking maneuvers, expanding the understanding of system behavior under complex operating conditions [18,19]. More recently, advanced CFD models have been developed to analyze complete braking systems, including the interaction between multiple components and heat dissipation in the presence of multiphase flows, allowing for a more realistic representation of the physical phenomenon [20]. Despite these advances, most studies have focused on disc analysis or global system configurations, while the specific influence of air duct geometry on flow distribution and heat transfer remains limited in the recent literature.

The findings reported in previous CFD investigations indicate that geometric optimization is one of the most effective strategies for improving brake cooling performance. For instance, García-León et al. [10] demonstrated that an appropriate airflow guidance system can significantly increase the convective heat transfer over the brake disc by promoting a more efficient air distribution. Similarly, Stauch et al. [12] showed that optimizing the cooling duct geometry

reduces aerodynamic losses while increasing the amount of cooling air reaching the braking system. These studies suggest that brake cooling efficiency depends not only on the airflow rate but also on how effectively the duct geometry controls flow development and pressure losses, highlighting the importance of geometric optimization in brake cooling system design.

Another fundamental parameter governing the thermo-fluid behavior of brake cooling systems is the Reynolds number, since it determines the flow regime and directly influences convective heat transfer. Li et al. [9], in a comprehensive review of ventilated brake discs, identified the Reynolds number as one of the dominant parameters controlling the cooling effectiveness of internal flow passages. Furthermore, Wang et al. [14] experimentally demonstrated that increasing the airflow through ventilated brake discs substantially enhances the convective heat transfer coefficient. Nevertheless, these studies also indicate that the influence of the Reynolds number cannot be analyzed independently because the duct geometry modifies flow acceleration, turbulence generation, and pressure losses, ultimately affecting the cooling performance.

These thermo-fluid mechanisms become particularly important in heavy-duty vehicles, where greater vehicle mass and prolonged braking events produce considerably higher thermal loads than those encountered in passenger cars. Yevtushenko et al. [4] demonstrated that repeated braking can lead to severe temperature accumulation, emphasizing the importance of efficient heat dissipation under demanding operating conditions. More recently, Orlandi et al. [20] developed an advanced CFD model for complete braking systems under transient conditions, providing valuable insight into brake thermal behavior. However, their study primarily focused on the overall braking system response rather than the isolated influence of brake cooling duct geometry. Consequently, the optimization of duct cross-sectional geometry for heavy-duty vehicles remains insufficiently addressed in the current literature.

Although previous studies have significantly improved the understanding of brake cooling systems, recent CFD investigations by Orlandi et al. [1] and Cravero and Marsano [16] have primarily focused on the thermal behavior of brake discs and complete braking system aerodynamics under realistic operating conditions. However, the isolated influence of brake cooling duct cross-sectional geometry on airflow distribution,

pressure losses, and the resulting convective heat transfer has received comparatively limited attention, particularly for heavy-duty bus applications. This knowledge gap limits the development of optimized duct designs capable of maximizing cooling performance while minimizing aerodynamic losses. Therefore, the present study aims to numerically evaluate the influence of circular and square duct cross-sections on the flow characteristics and thermal performance of a heavy-duty bus brake cooling system operating under straight-line braking conditions. Airflow behavior is first analyzed in terms of velocity distribution, flow patterns, and pressure losses for vehicle speeds of 60, 90, and 120 km/h. Subsequently, the resulting flow conditions are used to evaluate the convective cooling performance of a brake disc subjected to prescribed surface temperatures of 600, 700, and 800 K. The findings provide further insight into the relationship between duct geometry, airflow management, pressure losses, and heat dissipation, contributing to the design of more efficient brake cooling systems for heavy-duty vehicles.

METHODOLOGY

Description of the vehicle, the brake disc, and the duct

This study evaluates the flow characteristics inside a brake cooling duct and their influence on the thermal performance of a heavy-duty bus brake disc. The numerical procedure was carried

out in two successive stages. Initially, the airflow through the cooling duct was computed to determine the outlet flow conditions. These results were then used as the inlet boundary conditions for the thermal analysis of the brake disc under forced convection. The computational configuration adopted for both the duct and the brake disc is illustrated in Figure 1.

The computational domain consists of a brake cooling duct and a ventilated brake disc representative of a heavy-duty bus braking system. The cooling duct is formed by a wide frontal inlet that captures the incoming airflow and guides it through a curved passage toward the brake assembly. Both duct configurations have an overall length of approximately 780 mm, with an inlet section of 240×100 mm and an outlet located approximately 45 mm from the brake disc surface. The outlet geometry was defined as either a circular section with an internal diameter of 100 mm or a square section of 90×90 mm, providing nearly equivalent flow areas. Apart from the outlet cross-sectional geometry, all remaining duct dimensions and the flow path were maintained identical to ensure a consistent comparison between both configurations.

A standard ventilated brake disc commonly employed in heavy-duty buses was selected for the numerical analysis. The disc has an outer diameter of 420 mm, an inner diameter of 240 mm, a braking track width of 90 mm, and an overall thickness of 45 mm. Different braking conditions were represented by prescribing three uniform disc surface temperatures, allowing the

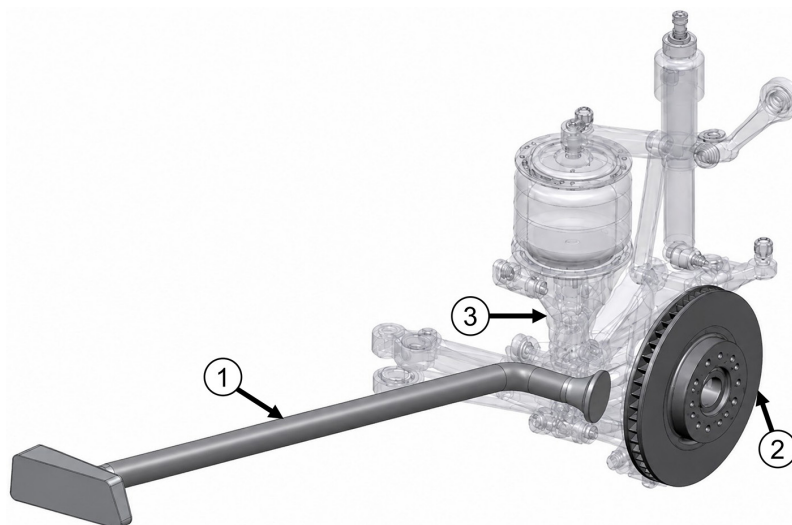


Figure 1. Schematic representation of the brake cooling system: (1) cooling air duct, (2) ventilated brake disc, and (3) suspension and steering assembly

influence of thermal loading on the cooling process to be evaluated. Rather than reproducing the complete vehicle geometry, the computational model preserves the actual relative position between the duct outlet and the brake disc so that the cooling airflow reaches the friction surfaces under representative operating conditions.

This study focuses on straight-line braking conditions, which represent one of the most critical operating scenarios for brake thermal load in heavy vehicles. Under these conditions, the outlet of the cooling duct remains aligned with the brake disc throughout the analysis. This modeling approach makes it possible to assess the influence of the duct geometry on the airflow delivered to the disc and its resulting cooling performance without introducing additional variables. Therefore, steering-induced wheel orientations were not considered in this analysis.

Numerical analysis

The flow and thermal behavior of the proposed brake cooling system were analyzed through three-dimensional CFD simulations carried out with ANSYS CFX. A steady-state formulation was adopted to solve the governing equations for both computational domains. Because the maximum air velocity remained below the compressibility limit ($\text{Mach} < 0.3$), the flow was treated as incompressible. Furthermore, the Reynolds numbers associated with the operating conditions and the duct configuration justified the use of a turbulent flow model throughout the numerical analysis.

The numerical solution was obtained by simultaneously solving the conservation equations of mass, momentum, and energy. Turbulence was represented using the SST $k-\omega$ model, which was adopted because of its capability to accurately capture near-wall flow behavior and regions subjected to adverse pressure gradients, both of which are expected within the curved cooling duct. The convective terms of the governing equations were discretized using the High Resolution scheme available in ANSYS CFX, whereas the diffusive terms were evaluated using second-order accurate spatial discretization. Pressure and velocity were solved simultaneously through the coupled solver implemented in ANSYS CFX. Convergence was assumed when the RMS residuals of all governing equations were below 10^{-5} , while the monitored mass flow rate and average outlet velocity remained unchanged between successive iterations.

The numerical procedure was divided into two separate simulation stages, each associated with a different computational domain. The first stage focused on the internal flow through the cooling duct, providing the outlet velocity and pressure characteristics required for the subsequent analysis. In the second stage, these flow conditions were applied to the brake disc model to investigate the heat transfer generated by forced convection.

The computational meshes for both numerical domains were generated using unstructured tetrahedral elements to accurately represent the complex geometry of the cooling duct and the ventilated brake disc. Local mesh refinement was applied in regions with pronounced geometric curvature and in areas where large velocity and thermal gradients were expected, particularly near the duct outlet, the brake disc surfaces, and the narrow gap between the duct and the disc. Additional refinement was also introduced close to the solid boundaries to improve the resolution of the near-wall flow, which is essential for the accurate prediction of turbulent transport using the SST $k-\omega$ model. The generated meshes were verified to ensure adequate element quality, avoiding highly distorted cells and providing stable numerical convergence throughout all simulations.

The airflow inside the cooling duct was analyzed under three inlet velocities corresponding to vehicle speeds of 60, 90, and 120 km/h, which were selected to represent typical urban, suburban, and highway operating conditions of heavy-duty buses. Air entered the computational domain with a uniform velocity profile at an inlet temperature of 300 K and a turbulence intensity of 5%, whereas the outlet was defined as an opening with a relative pressure of 0 Pa. A no-slip boundary condition was imposed on all solid surfaces, and the duct walls were assumed to be adiabatic by neglecting heat transfer through the wall material.

The outlet flow conditions obtained from the cooling duct simulations were subsequently used as the inlet boundary conditions for the brake disc model. To examine the effect of thermal loading, the brake disc surface was evaluated at three prescribed temperatures of 600 K, 700 K, and 800 K. These temperature levels represent progressively more severe braking conditions and were selected based on temperature ranges commonly reported in numerical investigations of brake cooling systems [12]. Since the objective of this study was to perform a comparative numerical evaluation of the

two duct geometries, no experimental measurements were conducted, and representative operating conditions from the literature were adopted. The rotational motion of the brake disc was not included in the numerical model so that the influence of the duct geometry on the forced-convection cooling process could be assessed independently while reducing computational cost. This simplification also ensures a direct comparison between the circular and square duct configurations under identical thermal boundary conditions.

Convergence analysis

The generated meshes consisted of unstructured tetrahedral elements with local refinement in regions of high curvature and expected large flow gradients. The mesh quality was verified prior to the simulations to avoid highly distorted elements and ensure numerical stability throughout the solution process. The sensitivity of the numerical solution to the mesh resolution was assessed before performing the simulations. Independent discretization analyses were carried out for both the cooling duct and the brake disc models using unstructured tetrahedral meshes. For each computational model, a series of progressively refined meshes was generated to identify a mesh density that provided stable numerical predictions while maintaining a reasonable computational cost.

Mesh refinement was continued until further increases in the number of elements produced only negligible changes in the selected monitoring variables. The convergence results for the circular and square cooling ducts are presented in Tables 1 and 2, respectively, for an inlet velocity equivalent to 90 km/h. The outlet velocity and the pressure drop were adopted as the evaluation criteria because they adequately describe the hydraulic behavior of the duct. Since the variation between consecutive mesh levels remained below 0.5% for both quantities, a mesh containing

approximately 600,000 elements was considered sufficient for the remaining simulations.

Table 3 summarizes the mesh convergence analysis carried out for the brake disc model. The boundary conditions correspond to the outlet flow obtained from the circular duct simulation at 90 km/h, which was selected as the reference case for the thermal calculations. The convergence criterion was established by monitoring the heat flux on the upper inner region of the brake disc and the total heat dissipation. For both variables, the variation between successive mesh refinements remained below 0.2%, indicating that additional mesh refinement would have a negligible influence on the numerical results.

The convergence analyses presented in Tables 1–3 demonstrate that further mesh refinement produces only negligible variations in the monitored variables for both computational models. Based on these results, meshes containing approximately 600,000 elements for the duct and 750,000 elements for the brake disc were adopted for the remaining simulations, providing an appropriate balance between solution accuracy and computational efficiency. Consequently, the differences observed between the duct configurations can be attributed to the geometric characteristics of the models rather than to discretization-related effects.

RESULTS AND DISCUSSION

This section presents the numerical results obtained from the flow and thermal analyses performed for the proposed brake cooling system. The analysis is divided into two parts. First, the influence of the duct geometry and the inlet airflow velocity on the flow field characteristics is evaluated using streamlines, outlet velocity, and pressure distributions. Subsequently, the thermal performance of the brake disc is investigated using the airflow conditions obtained from the

Table 1. Evaluation of the mesh independence of a circular section duct at 90 km/h

Elements	Outlet velocity (m/s)	Difference (%)	Pressure drop (Pa)	Difference (%)
201039	48.8672	-	9678.477	-
300077	49.6864	1.6847	8877.231	9.0259
400490	51.7001	3.8949	8471.771	4.7860
500887	51.8883	0.3627	8659.320	2.1659
600905	51.8847	0.0069	8677.982	0.2150
700473	51.8865	0.0035	8685.145	0.0825

Table 2. Evaluation of the mesh independence of a square section duct at 90 km/h

Elements	Outlet velocity (m/s)	Difference (%)	Pressure drop (Pa)	Difference (%)
201891	38.9805	-	7806.054	-
301384	38.5143	1.2104	7289.138	7.0916
400483	38.9902	1.2206	6695.716	8.8627
500659	39.3800	0.9898	6549.244	2.2365
600190	39.5219	0.3590	6464.084	1.3174
700456	39.4570	0.1645	6419.175	0.6996

Table 3. Evaluation of the mesh independence of the brake disc at 600K

Elements	Wall Heat Flux at inner top of disc (kW/m ²)	Difference (%)	Total Power dissipation (W)	Difference (%)
383448	58.19	-	6031.5	-
446034	55.89	4.1152	5173.4	16.5867
584110	53.87	3.7498	5498.7	5.9156
662144	52.91	1.8144	5718.7	3.8472
757585	52.46	0.8578	5812.9	1.6205
863267	52.17	0.5558	5822.1	0.1589

simulations, focusing on the heat flux distributions at the wall, the total heat dissipation, and the temperature fields of the air adjacent to the wall.

Effect of duct geometry and duct inlet velocity

Figures 2 and 3 show the velocity streamlines for circular and square ducts operating at inlet velocities corresponding to vehicle speeds of 60, 90, and 120 km/h, while Figures 4 and 5 show the associated pressure distributions. In all cases, the flow accelerates as it travels through the duct, reaching its maximum velocity in the elbow region and near the outlet section. This behavior is attributed to the combined effect of the duct curvature and the local flow redirection, which generates regions with high velocity gradients.

For the circular duct, the outlet velocity increases from 34.58 m/s at 60 km/h to 69.08 m/s at 120 km/h, while the square duct exhibits outlet velocities of 26.35 m/s and 52.66 m/s under the same operating conditions. Consequently, the circular configuration provides an outlet velocity approximately 31% higher than the square duct for all inlet conditions analyzed. This nearly constant difference suggests that the influence of duct geometry remains dominant throughout the analyzed velocity range.

The streamline patterns reveal significant differences in flow development between the two geometries. In the circular duct, the streamlines remain relatively smooth and uniformly distributed along the entire flow path, indicating a gradual redistribution of momentum as the air travels around the bend. In contrast, the square duct exhibits steeper local velocity gradients near the bend, where the abrupt change in flow direction, combined with the sharp corners, promotes flow non-uniformity. While large-scale separation is not observed, the concentration of streamlines near the bend indicates the presence of localized acceleration and greater momentum redistribution.

The pressure fields shown in Figures 4 and 5 complement these observations. As expected, the pressure decreases progressively from the inlet to the outlet as some of the available pressure energy is converted into kinetic energy. For the circular duct, the pressure drop increases from 4.06 kPa at 60 km/h to 14.87 kPa at 120 km/h. The corresponding values for the square duct are 2.96 kPa and 11.26 kPa, respectively. Therefore, the circular geometry consistently exhibits pressure losses approximately 32–37% greater than those of the square configuration.

The simultaneous increase in outlet velocity and pressure drop observed in the circular duct indicates a greater conversion of pressure energy

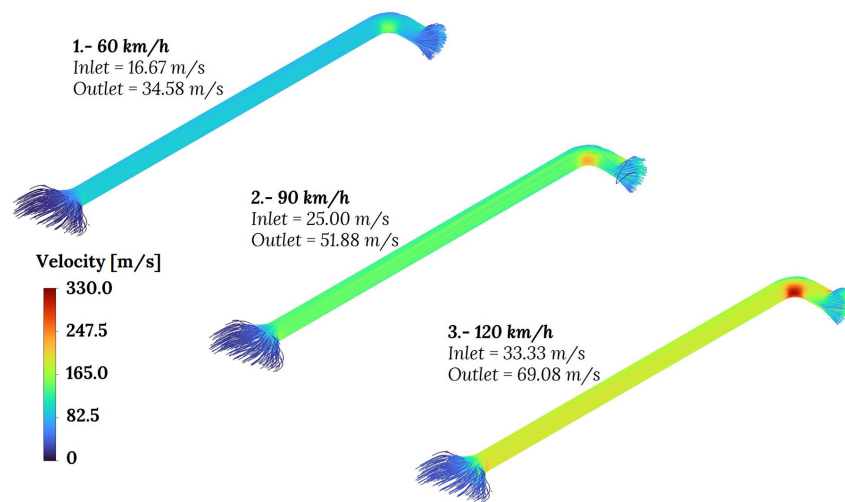


Figure 2. Velocity streamlines in the circular section duct for the three air inlet speeds: 60, 90 and 120 km/h

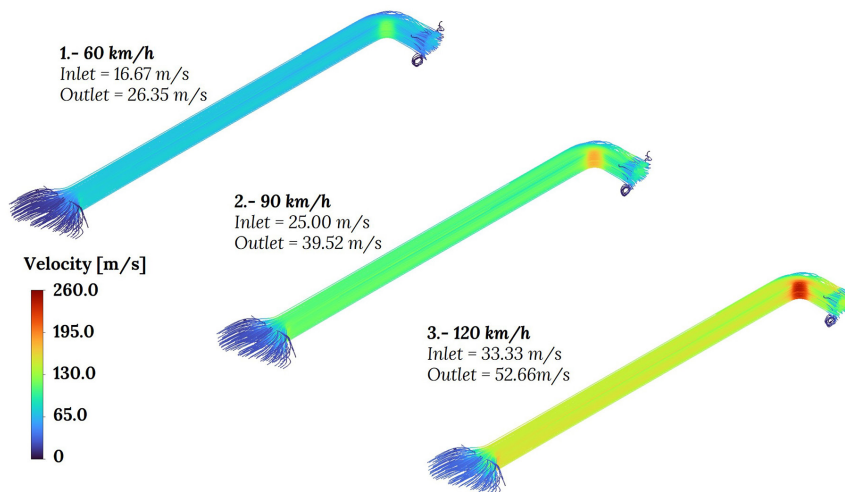


Figure 3. Velocity streamlines in the square section duct for the three air inlet velocities: 60, 90 and 120 km/h

into kinetic energy. However, this characteristic should not be interpreted as an inherent aerodynamic advantage. From a fluid mechanics perspective, greater pressure losses imply greater resistance to airflow, which may require higher driving pressure to maintain the same operating conditions. Consequently, the higher outlet velocity achieved with the circular duct comes at the cost of greater pressure loss.

From a fluid dynamics perspective, these results indicate that the cross-sectional geometry governs the balance between momentum transfer and hydraulic resistance inside the cooling duct. Although the circular duct experiences greater pressure losses, it also delivers a substantially higher outlet velocity, demonstrating a more efficient conversion of pressure energy into useful

airflow directed toward the brake disc. Therefore, the duct geometry should not be evaluated solely on the basis of minimizing pressure losses, but rather on its ability to maximize the cooling air supplied to the braking system.

From an energy perspective, the larger pressure drop observed in the circular duct represents an increased hydraulic resistance that requires a greater pressure difference to sustain the airflow. Nevertheless, this additional energy loss is compensated by a significant increase in the air momentum delivered to the brake disc, resulting in considerably higher convective heat transfer. Consequently, the energetic penalty associated with the pressure losses is justified when the primary design objective is to maximize brake cooling under severe operating conditions.

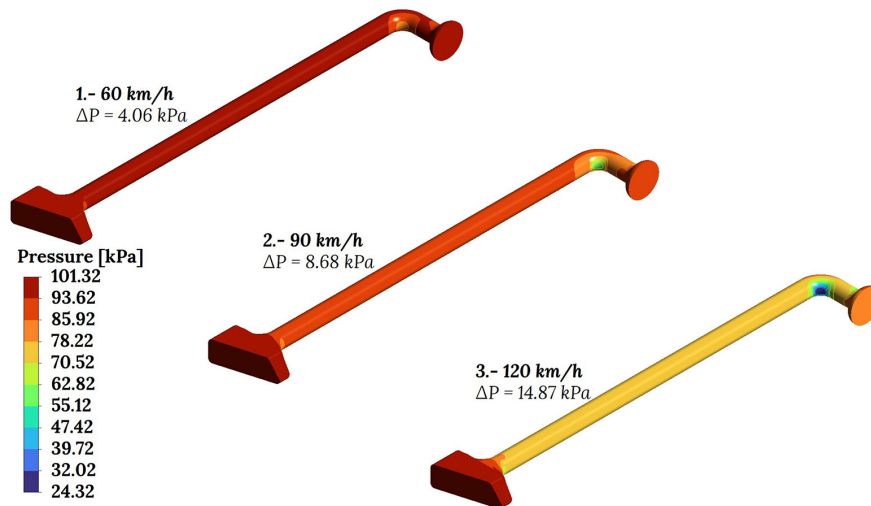


Figure 4. Absolute pressure distribution in the circular cross-section duct for the three air inlet speeds: 60, 90 and 120 km/h

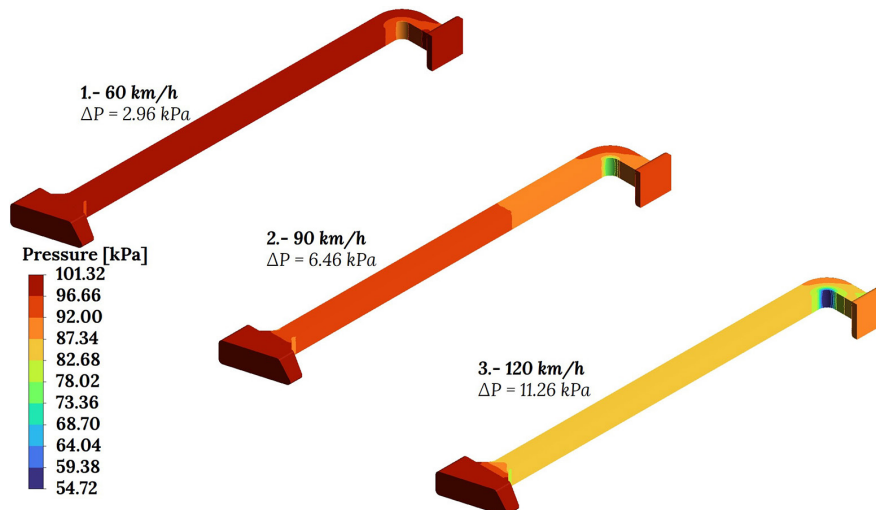


Figure 5. Absolute pressure distribution in the square section duct for the three air inlet speeds: 60, 90 and 120 km/h

An additional observation is that pressure losses increase approximately with the square of the inlet velocity. For example, when the vehicle speed increases from 60 to 120 km/h, the inlet velocity doubles, while the pressure drop increases by a factor of 3.66 in the circular duct and 3.80 in the square duct. This behavior is consistent with classical duct flow theory, where pressure losses are proportional to the dynamic pressure of the flow.

Overall, the results demonstrate that duct geometry significantly influences the balance between momentum transfer and pressure losses. The circular duct favors higher outlet velocities and a more uniform flow distribution, while the

square duct exhibits lower pressure losses but offers lower discharge velocities. Therefore, no configuration can be considered superior based solely on the flow characteristics obtained; the final evaluation must be based on subsequent thermal analysis, which assesses the actual cooling performance of the brake disc.

From an engineering standpoint, these findings suggest that the selection of the cooling duct geometry should be based on the thermal requirements of the braking system rather than exclusively on hydraulic performance. For heavy-duty buses operating under prolonged downhill braking or repeated braking cycles, the additional cooling capacity provided by the circular duct

may contribute to reducing brake fade, limiting thermal stresses, and extending the service life of brake components.

These findings are consistent with previous CFD studies reporting that the geometry of the cooling duct strongly influences the airflow distribution and pressure losses within brake cooling systems. García-León et al. [10] demonstrated that improved air guidance enhances convective heat transfer by increasing the airflow reaching the ventilated brake disc, while Stauch et al. [12] highlighted the importance of optimizing the cooling duct design to maximize cooling efficiency under different operating conditions. The lower pressure losses observed for the circular duct in the present study suggest a more efficient air delivery to the brake disc, which is beneficial for enhancing convective cooling performance.

Effect of disk surface temperature

Thermal simulations were performed using the outlet flow conditions obtained from the analysis presented in Section 3.1. For consistency, the air duct inlet velocity was set to 90 km/h, corresponding to an outlet velocity of 25.0 m/s. Under these conditions, the circular duct provided an outlet velocity of 51.88 m/s, while the square duct produced an outlet velocity of 39.52 m/s. These outlet velocities were subsequently imposed as inlet boundary conditions for the brake disc cooling simulations to evaluate the influence of duct geometry on convective heat transfer performance.

Figures 6 and 7 show the heat flux distributions at the wall obtained for the circular and square duct configurations, considering brake disc surface temperatures of 600 K, 700 K, and 800 K. The corresponding temperature distributions of the air adjacent to the wall are shown in Figures 8 and 9.

For both duct geometries, the increase in brake disc surface temperature results in a substantial increase in the amount of heat dissipated by the braking system. This behavior is expected, since the rate of convective heat transfer increases with the temperature difference between the hot disc surface and the incoming cooling air. Consequently, higher disc temperatures generate greater thermal gradients and higher rates of heat transfer across the entire braking surface.

For the circular duct configuration, the total thermal power dissipated increases from 5812.9 W at 600 K to 7770.1 W at 700 K and reaches 9727.3 W at 800 K. Relative to the 600 K condition, these values represent increases of 33.7% and 67.3%, respectively. Similarly, the square duct exhibits power dissipation values of 3851.8 W, 5148.7 W, and 6445.6 W for disk surface temperatures of 600 K, 700 K, and 800 K, respectively.

A direct comparison between the two geometries reveals a significant influence of the duct shape on cooling efficiency. At a disk temperature of 600 K, the circular duct dissipates 1961 W more thermal energy than the square duct, representing a 50.9% improvement. At 700 K and 800 K, the additional heat dissipation increases to 2621 W

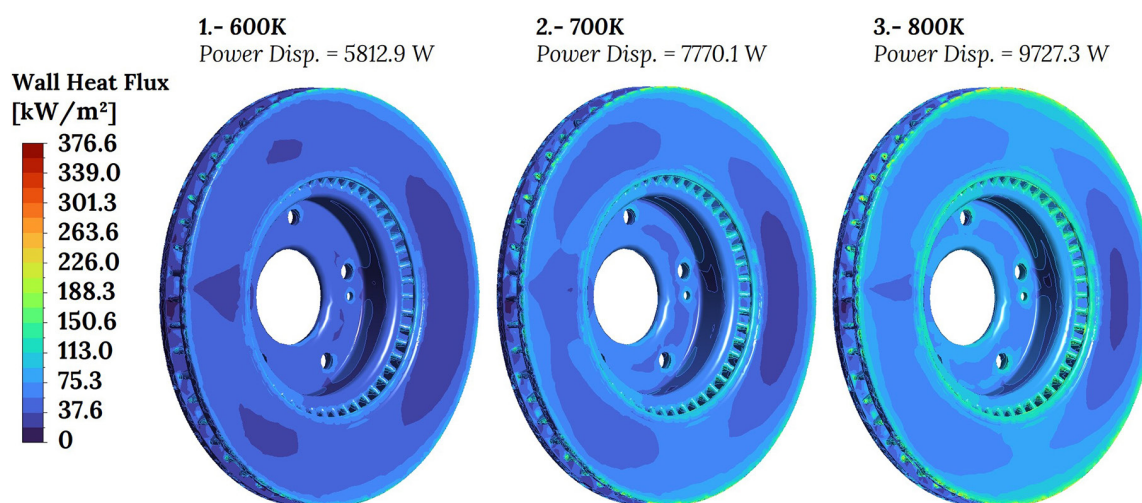


Figure 6. Heat flux distribution on the wall along the brake disc with the circular section duct, considering the three disc surface temperatures: 600 K, 700 K and 800 K

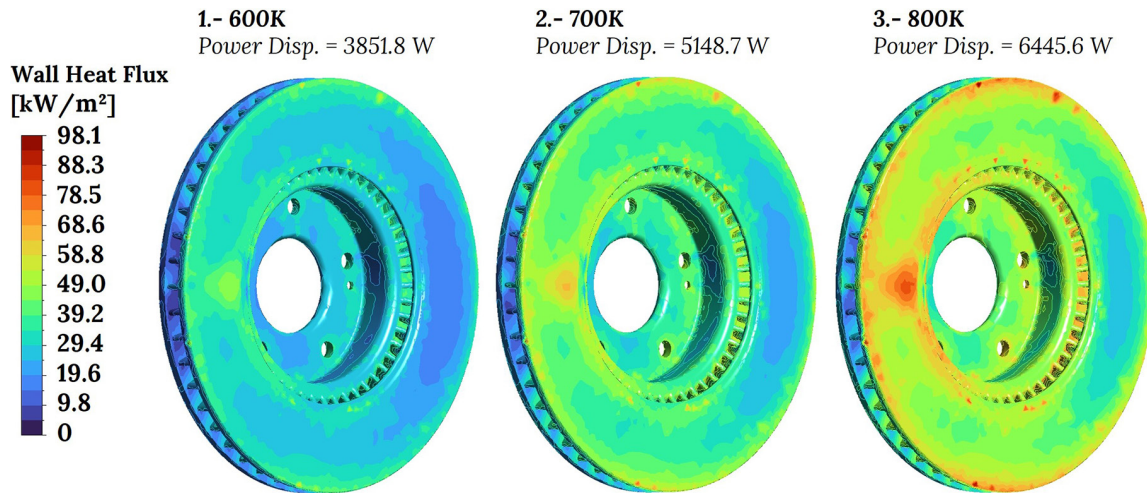


Figure 7. Heat flux distribution on the wall along the brake disc with the square section duct, considering the three disc surface temperatures: 600 K, 700 K and 800 K

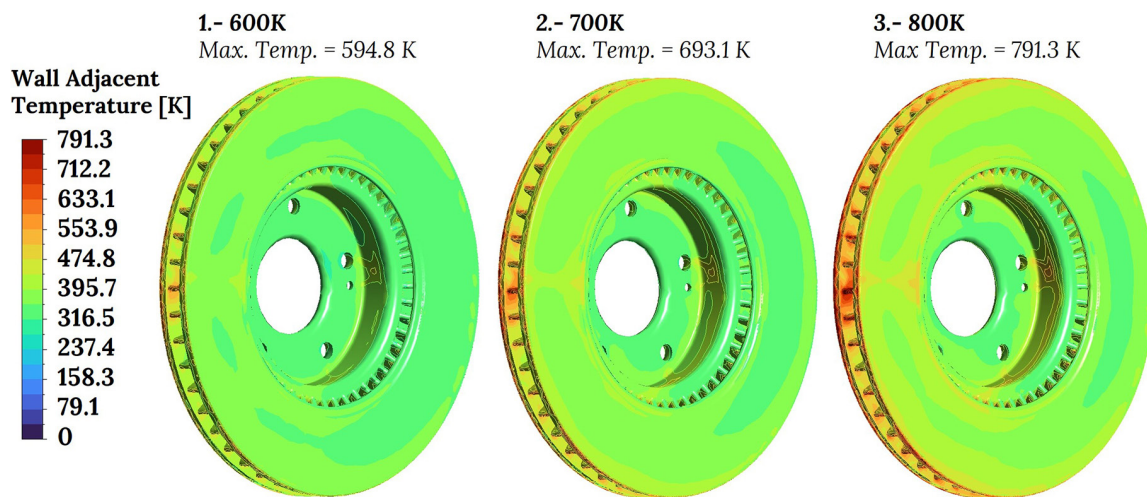


Figure 8. Temperature distribution adjacent to the wall along the brake disc with the circular section duct, considering the three disc surface temperatures: 600 K, 700 K and 800 K

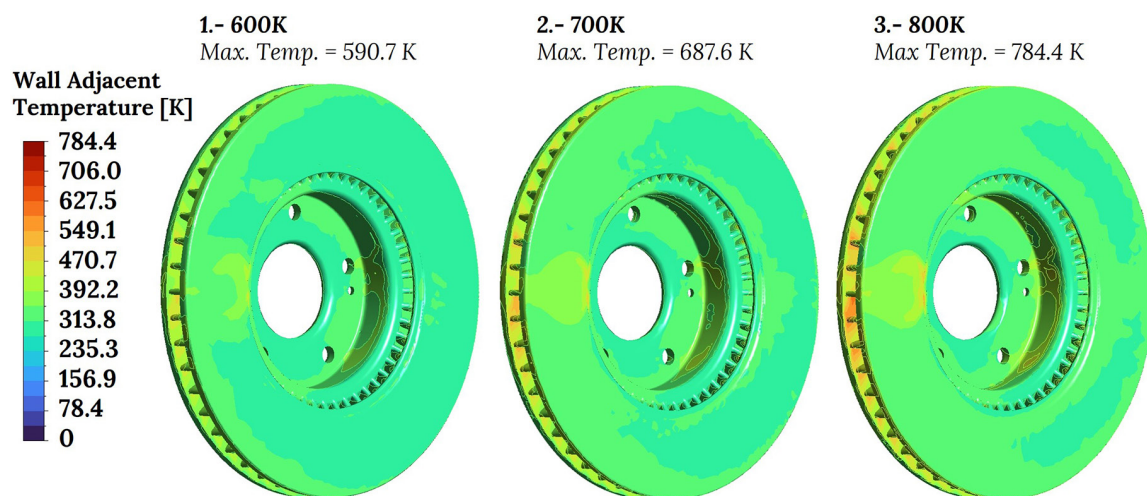


Figure 9. Distribution of the temperature adjacent to the wall along the brake disc with the square section duct, considering the three disc surface temperatures: 600 K, 700 K and 800 K

and 3282 W, respectively, maintaining the same relative improvement of approximately 51%. The persistence of this difference across all temperatures studied suggests that the flow characteristics determined by the duct geometry remain the dominant factor governing thermal performance.

The heat flow contours on the wall, shown in Figures 6 and 7, provide further information about this behavior. In the case of the circular duct, the regions of high heat flow are distributed more evenly along the outer friction ring and throughout all the ventilation channels. The heat transfer pattern appears relatively homogeneous, indicating that the incoming airflow interacts effectively with a larger fraction of the available heat transfer area. In contrast, the square duct exhibits a less uniform distribution, characterized by localized regions of high intensity surrounded by larger areas with comparatively lower heat flow. This behavior indicates that the cooling airflow is not distributed with the same efficiency across the disc surface, reducing the overall heat removal capacity.

An important observation is that the approximately 31.3% increase in cooling air velocity provided by the circular duct results in a substantially greater improvement in thermal performance, achieving nearly 51% higher heat dissipation. This finding suggests that the benefits of the circular configuration are not solely associated with increased airflow momentum, but also with improved airflow distribution across the brake disc surface. Consequently, a larger portion of the disc actively participates in the convective heat transfer process.

These results indicate that the improvement in thermal performance cannot be explained solely by the increase in outlet velocity. Instead, the circular duct promotes a more homogeneous airflow distribution over the brake disc surface, increasing the effective area participating in convective heat transfer. This demonstrates that duct geometry influences not only the magnitude of the airflow but also the spatial distribution of cooling, which ultimately governs the overall heat dissipation capacity.

The temperature distributions of the air adjacent to the wall, shown in Figures 8 and 9, complement the heat flow analysis. It should be noted that these contours represent the temperature of the fluid immediately adjacent to the wall, rather than the temperature of the brake disc surface itself. Therefore, they provide information about

the thermal response of the cooling airflow after receiving heat from the disc.

For the circular duct configuration, the maximum temperature adjacent to the wall increases from 594.8 K at a disk surface temperature of 600 K to 693.1 K and 791.3 K for disk temperatures of 700 K and 800 K, respectively. The corresponding values for the square duct are 590.7 K, 687.6 K, and 784.4 K. As expected, higher disk temperatures result in higher temperatures in the thermal boundary layer adjacent to the wall.

Interestingly, despite the significant differences observed in total heat dissipation, the maximum temperatures adjacent to the wall remain relatively similar for both duct geometries. The differences are only 4.1 K, 5.5 K, and 6.9 K for disk temperatures of 600 K, 700 K, and 800 K, respectively, corresponding to deviations of less than 1% in all cases. This result indicates that the superior thermal performance of the circular duct is not primarily associated with a substantial reduction in the local fluid temperature, but rather with a more efficient distribution of the cooling airflow and a higher rate of convective heat transfer across the disc surface. This demonstrates that the spatial distribution of the cooling flow is more decisive than the maximum local temperature of the air adjacent to the wall in characterizing the overall cooling performance of the system.

Overall, the results demonstrate that the brake disc temperature significantly influences the magnitude of heat transfer, while the duct geometry determines the efficiency with which the available cooling air is utilized. Under all the thermal conditions studied, the circular duct consistently provides higher heat dissipation rates and more uniform heat transfer distributions than the square duct, confirming that the flow characteristics have a direct impact on the thermal behavior of the brake cooling system.

The enhanced cooling performance obtained with the circular duct configuration agrees with previous numerical and experimental investigations emphasizing the dominant role of airflow management in brake thermal control. Orlandi et al. [1] reported that the effectiveness of brake cooling strongly depends on the airflow distribution surrounding the brake disc, while Sathiyar et al. [11] experimentally demonstrated that increasing the air mass flow through ventilated brake discs significantly enhances heat dissipation. Likewise, Li et al. [9] concluded that optimizing the airflow path is one of the most

effective strategies for improving convective cooling. The approximately 51% increase in heat dissipation obtained in the present work further confirms that duct geometry plays a decisive role in maximizing the thermal performance of heavy-duty braking systems.

For heavy-duty vehicles, improving brake cooling performance has direct practical implications because it contributes to maintaining braking efficiency during prolonged service conditions characterized by repeated or sustained braking events. Enhanced heat dissipation reduces the likelihood of brake fade and excessive thermal loading, thereby improving system reliability while potentially reducing maintenance requirements associated with thermal degradation of friction components.

CONCLUSIONS

A numerical study was conducted to evaluate the influence of duct cross-sectional geometry on the flow characteristics and brake cooling performance of a heavy-duty bus. The results demonstrated that the cooling duct geometry significantly affects airflow development and the consequent convective cooling capacity of the brake disc. The circular duct consistently generated higher exit velocities than the square duct at all vehicle speeds tested. At 90 km/h, the exit velocity reached 51.88 m/s for the circular configuration, compared to 39.52 m/s for the square configuration, corresponding to an increase of approximately 31%. However, this improvement was accompanied by a greater pressure drop, indicating a trade-off between acceleration and flow resistance. Thermal simulations revealed that the higher airflow supplied by the circular duct substantially improved the brake disc's heat dissipation capacity. At a surface temperature of 600 K, the total power dissipated was approximately 51% higher than that obtained with the square duct. Similar behavior was observed for thermal conditions of 700 K and 800 K, confirming the superior cooling efficiency of the circular configuration across the entire range of operating conditions considered. Although both duct geometries produced comparable peak air temperatures adjacent to the wall, the circular duct achieved greater overall heat dissipation. This result indicates that the distribution and momentum of the cooling airflow play a more significant

role in brake cooling performance than the local peak air temperature near the disc surface. Overall, the results demonstrate that the cross-sectional geometry of the cooling duct is a key design parameter in brake thermal management systems. Among the configurations evaluated, the circular duct provided the most effective cooling performance and thus represents a promising solution for improving brake cooling in heavy-duty buses operating under severe thermal conditions.

The present study was limited to steady-state CFD simulations under prescribed brake disc temperatures and straight-line braking conditions, without considering brake disc rotation, transient braking events, or experimental validation. Despite these simplifications, the numerical results provide useful engineering insight into the influence of duct geometry on brake cooling performance. From an industrial perspective, the proposed circular duct configuration could be incorporated into the design of heavy-duty bus braking systems to improve thermal management, reduce the risk of brake fade, and potentially extend the service life of braking components. Future work should include transient simulations, brake disc rotation, experimental validation, and optimization of duct geometry under realistic vehicle operating conditions to further support the development of more efficient brake cooling systems.

REFERENCES

1. Orlandi F., Coccetta M., Meleti S., Milani M., Montorsi L. A comprehensive CFD approach of a thermal analysis of a braking discs system under different design conditions. *International Journal of Thermofluids* 2025;25:101004. <https://doi.org/10.1016/j.ijft.2024.101004>
2. Vrabel J., Jagelcak J., Zamecnik J., Caban J. Influence of emergency braking on changes of the axle load of vehicles transporting solid bulk substrates. *Procedia Engineering* 2017;187:89–99. <https://doi.org/10.1016/j.proeng.2017.04.354>
3. Jančár A., Ondruš J., Macurová L., Caban J. Alternative devices for measuring the braking characteristics of vehicles. *Automotive Safety* 2024 2025:249–57. <https://doi.org/10.1201/9781003465959-20>
4. Yevtushenko A., Topczewska K., Kuciej M. Analytical determination of the brake temperature mode during repetitive short-term braking. *Materials* 2021;14:1912. <https://doi.org/10.3390/ma14081912>
5. Sha Z., Lu J., Hao Q., Yin J., Liu Y., Zhang S. Numerical simulation of heat production and

- dissipation of ventilated brake disc for high-speed trains under the action of the flow field. *Applied Sciences* 2022;12:10739. <https://doi.org/10.3390/app122110739>
6. Rybicka I., Caban J., Vrabel J., Sarkan B., Stopka O., Misztal W. Analysis of the safety systems damage on the example of a suburban transport enterprise. 2018 XI International Science-Technical Conference Automotive Safety 2018:1–6. <https://doi.org/10.1109/autosafe.2018.8373323>
7. Caban J., Vrabel J., Šarkan B., Kuranc A., Słowik T. Operational tests of brake fluid in passenger cars. *Period Polytech Transp Eng* 2020;49:126–31. <https://doi.org/10.3311/pptr.14583>
8. Boryga M., Wrzesiński B. Experimental investigation of single flare in brake tubes. *Agricultural Engineering* 2026;30:49–61. <https://doi.org/10.2478/agriceng-2026-0003>
9. Li J., Tao L., Gu J.L., Chen C., Chen Y. Review on convective heat transfer in internal channel of ventilated brake disc of vehicle. *Journal of Traffic and Transportation Engineering*. 2022 Apr;22(2):19-40. <https://doi.org/10.19818/j.cnki.1671-1637.2022.02.002>
10. García-León R.A., Afanador-García N., Gómez-Camperos J.A. Numerical study of heat transfer and speed air flow on performance of an auto-ventilated disc brake. *Fluids* 2021;6:160. <https://doi.org/10.3390/fluids6040160>
11. Sathiyar T.K., Vdovin A., Johansson S., Sebben S. Experimental thermal analysis of brake cooling relative to mass flow of air through a ventilated brake disc. *Case Studies in Thermal Engineering* 2024;64:105530. <https://doi.org/10.1016/j.csite.2024.105530>
12. Stauch R., Bög J., Grabant S., König R., Traub D. Analysis of the cooling concept of the braking system of a formula student racing car using CFD simulation and 1D simulation. *DDF* 2021;412:115–29. <https://doi.org/10.4028/www.scientific.net/ddf.412.115>
13. Sha Z., Lu J., Hao Q., Yin J., Liu Y., Zhang S. Numerical simulation of heat production and dissipation of ventilated brake disc for high-speed trains under the action of the flow field. *Applied Sciences* 2022;12:10739. <https://doi.org/10.3390/app122110739>
14. Wang L., Lu X., Li H., Gong W., Wang L. A new experimental method to study the convective heat transfer characteristics of the interior passages of ventilated disc brakes. *International Journal of Thermal Sciences* 2022;179:107675. <https://doi.org/10.1016/j.ijthermalsci.2022.107675>
15. Chaudhary A.S., Ghodake P., Khewale S. Coupled CFD Simulation of brake duty cycle for brake system design. *SAE Technical Paper Series* 2021;1. <https://doi.org/10.4271/2021-26-0360>
16. Cravero C., Marsano D. Flow and thermal analysis of a racing car braking system. *Energies* 2022;15:2934. <https://doi.org/10.3390/en15082934>
17. Tito D.N., Suarez D.Q., Gutierrez J.A., Quispe W.B., Carpio P.P., Arapa C.D. Design and implementation of fog light ducts to optimize hot air recirculation and heat transfer in the Toyota Hilux. *IJHT* 2025;43:1631–43. <https://doi.org/10.18280/ijht.430502>
18. Broniszewski J., Piechna J.R. Fluid-structure interaction analysis of a competitive car during brake-in-turn manoeuvre. *Energies* 2022;15:2917. <https://doi.org/10.3390/en15082917>
19. Yang K., Guan X., Zhang X., Liu C. CFD-aided approach of modelling and dynamic characteristic optimization for a highly nonlinear auxiliary braking system. *Engineering Applications of Computational Fluid Mechanics* 2022;16:1546–66. <https://doi.org/10.1080/19942060.2022.2103589>
20. Orlandi F., De Marco C., Meleti S., Manian A., Milani M., Montorsi L. CFD analysis of the main operating parameters for a complete braking friction system under transient working conditions. *International Journal of Thermofluids* 2025;26:101100. <https://doi.org/10.1016/j.ijft.2025.101100>