

Enhancing Nmax Gen 2 Brake Cooling Through CFD-Based Duct Profile Optimization

Teng Sutrisno^{1*}, Marcell Chevilino¹, Ninuk Jonoadji¹, Ian Hardianto Siahaan¹,
Amelia Sugondo¹, Willyanto Anggono¹, Roche Alimin¹, Melvin E. Simanjuntak²,
Micheal Suryajaya³

¹ Program Studi Teknik Mesin, Fakultas Teknologi Industri, Petra Christian University
Jl. Siwalankerto 121-131, Surabaya 60236, Indonesia

² Program Studi Teknik Mesin, Politeknik Negeri Medan

Jl. Almamater No: 1, Padang Bulan, Medan, Sumatera Utara 20111, Indonesia

³ Mechanical-Aerospace-Automotive Engineering Department, Coventry University
Priory Street, Coventry United Kingdom, CV1 5FB, United Kingdom

*Corresponding author; E-mail: tengsutrisno@petra.ac.id

ABSTRAK

Upaya peningkatan performa sistem pengereman sepeda motor menjadi salah satu upaya peningkatan keselamatan berkendara, terutama pada kondisi kecepatan tinggi. Tujuan utama penelitian ini adalah mengevaluasi pengaruh penggunaan brake duct dalam pendinginan piringan cakram dengan menggunakan metode computational fluid dynamics (CFD). Penelitian ini juga bertujuan untuk menentukan desain brake duct yang paling optimal untuk digunakan pada kendaraan sepeda motor. Penentuan keakuratan hasil simulasi, dilakukan eksperimen dengan lorong angin untuk menentukan besar error antara hasil simulasi dan eksperimen. Hasil validasi menunjukkan bahwa besar error yang dimiliki oleh simulasi sebesar 5,64% kondisi sudah cukup baik. Simulasi dilakukan dengan memodelkan sistem pengereman sepeda motor dan brake duct, kemudian dianalisis menggunakan ANSYS Fluent. Untuk menentukan konfigurasi brake duct yang paling optimal, dilakukan variasi dimensi radius bagian dalam, panjang, dan ukuran diffuser. Konfigurasi yang paling baik adalah konfigurasi yang menghasilkan brake duct dengan pressure loss paling rendah, dan peningkatan tekanan statis paling tinggi. Hasil simulasi menunjukkan bahwa konfigurasi terbaik untuk diterapkan adalah konfigurasi R 20mm, L180mm dan D 8,75mm x10 mm.

Kata Kunci: brake duct, computational fluid dynamics (CFD), pendinginan rem sepeda motor.

ABSTRACT

Efforts to improve the performance of motorcycle brake system performance are essential for enhancing riding safety, particularly at high speeds. The main objective of this study is to evaluate the effect of a brake duct on brake disc cooling using computational fluid dynamics (CFD). This study also aims to determine the optimal brake duct design for motorcycles applications. To assess the accuracy of the numerical simulations, wind tunnel experiments were conducted to quantify the error between the simulation and experimental results. The validation results showed that the simulation error was 5.64%, indicating good agreement with the experimental data. The simulation was performed by modelling the motorcycle braking system and brake duct, then analyzed using ANSYS Fluent. To identify the optimal brake duct configuration, variations in the inner radius R, duct length L, and diffuser size D were evaluated. The best configuration was defined as the design that produced the lowest pressure loss and the highest ΔP statis. The simulation results indicate that the optimal configuration corresponds to R = 20 mm, L = 180 mm, and D = 8.75mmx 10mm.

Keywords: brake duct, computational fluid dynamics (CFD), cooling system motorcycle.

INTRODUCTION

In 2022, over 5,200 accidents in Indonesia were caused by brake failures [1]. If your brakes are bad, it will be harder to stop and maneuver the car, thus increasing the risk of an accident. Disc

brakes are widely used on motorcycles in Indonesia, particularly on the front wheel. Friction in disc braking systems causes a significant rise in temperature during operation. When the temperature of the disc brake exceeds its operational limit, both the brake pads and disc rotor become less effective, and the pads lose their grip. The pads can typically operate at temperatures up to 180°C [2]. However, if the temperature increases above this level, the coefficient of friction can decline by up to 16.7%, resulting in increased brake force requirements, longer stopping time, and greater stopping distance [3]. Disc rotors can also fail as they expand and develop uneven surfaces at high temperatures. Therefore, improving the cooling performance of the braking system by changing wheel shape, disc shape, or adding more parts. A well-designed wheel can provide air to the brake system [4], and vented discs can increase airflow and reduce turbulence, thereby lowering disc brake temperature [5]. The study says that ventilation in disc brakes has a big effect on how heat moves and how air flows. Other parts, such as vortex generators [6], forced cooling mechanisms, brake ducts, can also improve the cooling system of disc brakes.

A brake duct is an air channel component that directs airflow toward the brake system. This component enhances cooling by directing increased airflow toward the braking system [7]. A patent system that "Active Brake Cooling Ducts" can significantly improve heat transfer in the braking system. Siddharth Singh *et al.* [8] have demonstrated, using finite element model, that the design of a brake disk with curved ventilation, holes, and slots can determine its stress and temperature distributions. Their findings provide valuable insight into the structural and thermal characteristics of brake discs, which heavily depend on their geometric shape. Therefore, the design optimization process can be supported by the finite volume method, which enables accurate; this method is known as computational fluid dynamics (CFD).

The numerical study by Zheng *et al.* [9] was shown to be consistent with experiments. Braked on a vehicle at high speeds in a closed high-speed train system reached a maximum disk brake temperature of 1,137.62 K in their study. Their work simulation method can help provide a reference for temperature distribution around the brake disc when experimental conditions are difficult to achieve. This proves that simulation can be used as an alternative to exploring high-speed objects. In addition, Marin *et al.* [10] demonstrated that CFD can predict the airflow cooling of car brakes, which was later validated in wind tunnel experiments. In the CFD modelling specifically for brake system modeling, the simulation method, turbulence model, and software package used for the simulation need to be considered.

According to Tripathi *et al.* [11], the temperature of a disc brake is highly dependent on the proper rotor design and the selection of materials with high heat-dissipation capabilities, which provide better performance during braking. Disc brakes used in the automotive industry must exhibit uniform stress distribution, effective temperature management, and adequate safety factors. Their study reported that the titanium alloy TI550 demonstrated better results compared to gray cast iron. In addition, ventilated disc brakes are structurally safe as the total maximum stress remains within the material's ultimate stress limits. Furthermore, disc brakes can be used in vehicles if the deformation ranges between 0.00113 and 0.000973 mm for solid and ventilated disc brakes. The maximum temperature and overall heat flux of ventilated disc brakes are slightly lower. This is due to the larger surface area available for heat dissipation.

Vehicle braking systems are crucial for road safety, especially under high-temperature conditions. The temperature rise in disc brakes is caused by friction between the disc and the brake pads, which can reduce brake efficiency. Continuous braking, such as when descending steep slopes at high speeds, may significantly elevate brake temperature, potentially leading to brake failure. Approximately 30% of accidents are associated with brake-related issues, with thermal effects identified as a major contributing factor [12]. This phenomenon is always marked by squeaking sounds from the brakes before they malfunction [13]. Although continuous technological advancements have improved brake system noise characteristics, further research is still required to enhance thermal performance.

One approach to reducing disc brake temperature is optimizing the disc brake's ventilation design, as carried out by Li and Yang [14]. Using the Taguchi optimization method, the study analyzed three parameters: number of fins, fin thickness, and blade angle. The optimized design increased the ventilation gap from 10 mm to 14 mm and the blade bend angle from 29° to 37°. The cooling

performance improved by 6.1% and 8.8%, respectively, during a 100-second heat dissipation period following emergency braking at a vehicle speed of 60 km/h. Agrawal *et al.* [15] reported that the thermal stability of brake system performance is significantly affected by rotational speed and load. The experimental results show that the disc brake temperature ranges between 97.8°C and 306.7°C in rotational speeds of 300-1100 rpm with loads from 20-80 N. CFD simulation has predicted and validated the disc brake temperature to be between 36.45 and 91.686°C. This shows that the CFD method has been widely used in disc brake cooling.

Another effort by Dewanto *et al.* [16] involved adding an active cooling system to the disc brakes when they experience overheating. The main problem with four-wheeled vehicles equipped with disc brakes is the potential for excessive heat, which can reduce braking efficiency. Therefore, an innovative solution was developed with an active cooling system controlled by an Arduino Nano microcontroller. Testing results showed that this system is highly effective at preventing overheating and has proven to work stably under all conditions, with the added advantage of standardized mechanisms and control systems. Although the initial performance was very good, the research results indicate that to handle much heavier braking conditions, it is necessary to increase the capacity of the water storage tank and pump to maintain optimal cooling.

High-powered bicycle disc braking system can generate extreme temperatures that can cause brake fluid boiling and performance degrade [17]. This has prompted research into the friction performance and thermal behavior of bicycle disc brakes by testing various types of brake pads, namely sintered metallic, organic, and 'power' organic. The coefficient of friction is highly dependent on temperature and normal force, with organic pads exhibiting higher friction. Using the MATLAB model, disc temperatures were successfully predicted during field tests based on friction data. The core of this research presents a comprehensive methodology that combines dynamometer testing, numerical modeling, and field-test data, including elevation history and line pressure, to accurately predict brake temperatures and transient friction performance.

Swirl flow energy is a form of flow energy that is significant beyond the conventional kinetic energy generated by linear velocity, as reported by Simanjuntak *et al.* [18],[19]. Demonstrating this phenomenon can substantially enhance heat transfer effectiveness. Swirl flow enhances heat transfer by disrupting the thermal boundary layer near the surface through strong turbulence induction, while also extending the fluid's path and contact time with the hot surface. This has been proven through a comprehensive methodology, namely direct experiments and validation with mathematical models. The results indicate that manipulating the flow pattern into a swirl is a highly effective strategy to improve thermal efficiency in engineering systems without increasing axial velocity, which can lead to high-pressure losses. In addition, support for turbulence energy, defined as turbulence kinetic energy associated with swirl flow, can increase the flow's energy. This evidence is consistent with the results reported by Sutrisno *et al.* [20].

Research in air flow simulation across brake ducts has been conducted by Raja *et al.* [21]. The study presented a simulation of brake duct air flow and its impact on the braking system's temperature. The study modeled a nozzle directing airflow toward a heated brake disc subjected to a heat flux of 120.084 W/m². The inlet airflow velocities were 27.7 m/s and 13.85 m/s. Simulation results indicated significant acceleration of airflow within the duct, reaching average velocities of 121.143 m/s and 60.123 m/s, respectively. These enhanced airflow conditions contributed to a reduction in brake disc surface temperature and influenced the outlet air temperature from the duct.

This study performs a fluid flow simulation of a brake duct to evaluate its effectiveness in improving the cooling performance of a motorcycle braking system. Enhanced cooling is expected to reduce brake temperature and mitigate performance degradation. The objective of this research is to develop an optimized duct geometry capable of reducing disc brake temperature by up to 30%. Specifically, variations in inner radius, duct length, and diffuser size are investigated to determine the most effective configuration.

acceleration of 9.81 m/s^2 , with a downward vector applied to represent real operating conditions. To model the complex turbulence phenomena induced by disc rotation, renormalization group (RNG) $k-\epsilon$ turbulence model is used. The selection of the RANS model is motivated by its enhanced capability to account for rotational effects, flow separation, and streamline curvature through mathematically derived model constants. The working fluid is air at room temperature ($25 \text{ }^\circ\text{C}$). The governing equations are solved using the semi-implicit method for pressure-linked equations-consistent (SIMPLEC) algorithm, chosen for its stability and robustness in internal flow simulations. Spatial discretization of pressure, momentum, turbulent kinetic energy (k), and dissipation rate (ϵ) is performed using a first-order scheme. Although simple, choosing the first order provides higher numerical stability and faster convergence. Finally, the boundary conditions at the outlet section are set to outflow to model the fully existing airflow in the brake duct system. The convergence criterion for the residuals is set to 10^{-5} .

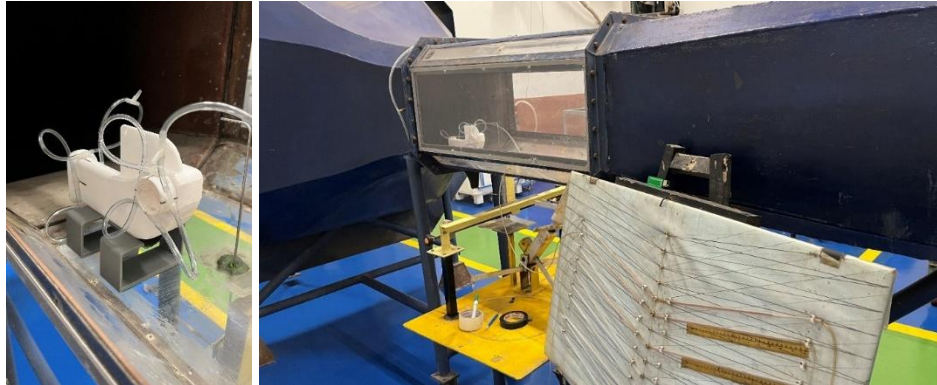


Figure 3. NMax Ducting validation process with a wind tunnel

Table 1. Measurement results in the wind tunnel

Inlet velocity at ducting	Outlet velocity at ducting	Inlet Statis Pressure at ducting	Outlet Statis Pressure at ducting
V_{in} [m/s]	V_{out} [m/s]	$P_{statis in}$ [Pa]	$P_{statis out}$ [Pa]
5.53	18.20	101305.90	101324.93
5.97	18.76	101302.70	101324.93
7.14	19.16	101293.10	101324.92
7.82	18.89	101286.70	101324.93

Table 2. Comparison of experiments and numerical result

<i>Error CFD</i>					
EXPERIMENT		CFD		ERROR	
V_{in} [m/s]	V_{out} [m/s]	V_{in} [m/s]	V_{out} [m/s]	IN [%]	OUT [%]
7.82	18.89	7.88	17.83	0.825	5.642

RESULTS AND DISCUSSION

This research optimized the inlet shroud curvature (R) as shown in Figure 4. The results indicate an inverse relationship between the inlet shroud radius of curvature (R) and pressure loss. In the configuration with $R = 0 \text{ mm}$, the system exhibited the highest resistance (1672.13 Pa), as the sharp corner forced the fluid to undergo an abrupt directional change, leading to increased turbulence and energy loss. When the radius was increased to 10 mm , the pressure loss decreased to 1549.69 Pa . The lowest pressure loss (1381.76 Pa) was achieved with $R = 20 \text{ mm}$, representing the optimal configuration among the cases considered. Overall, increasing the radius from 0 mm to 20 mm resulted in a pressure loss reduction of approximately 17.4% . This improvement can be attributed to the enhanced aerodynamic characteristics of the inlet channel geometry. Sharp corners promote flow separation and vortex formation, which disrupt the main flow and increase

energy losses. By smoothing the inlet shroud-curvature (R) up to 20 mm, the fluid can follow the contour of the walls more 'smoothly,' allowing the boundary layer to remain attached to the surface and minimizing unnecessary friction. Therefore, the R = 20 mm design is highly recommended if energy efficiency is a top priority, as it directly reduces the workload on the system's driving engine.

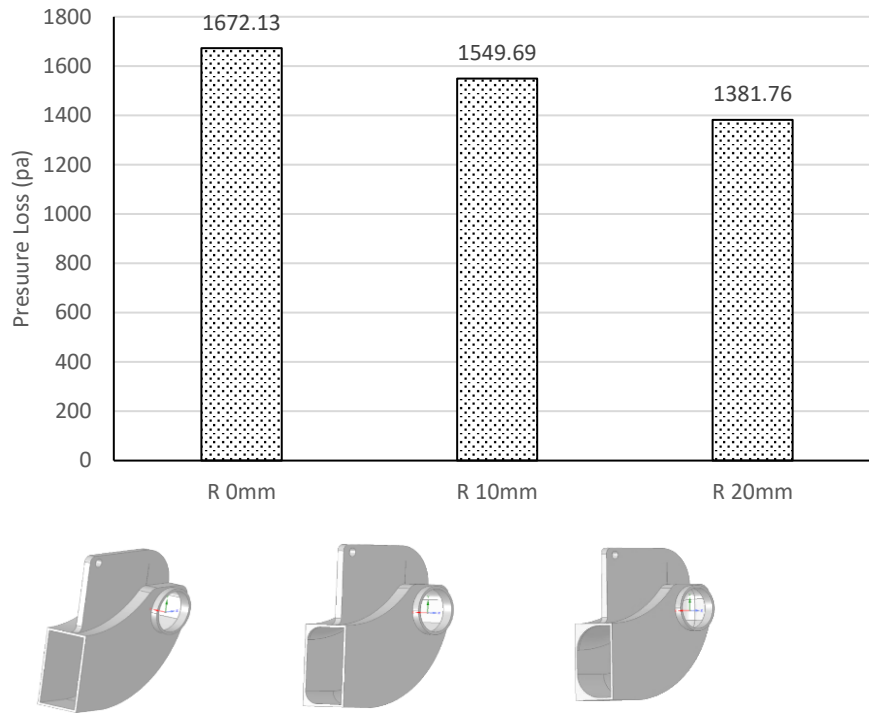


Figure 4. Pressure losses in various inlet shroud radius of curvature (R)

As shown in Figure 5, the pressure loss (P_{loss}) in the model, while the dynamic pressure difference ($\Delta P_{dynamic}$) represents the increase in velocity due to the diffuser cross-sectional variation. The ratio between $\Delta P_{dynamic}$ and P_{loss} reflects the relative conversion of energy into kinetic energy compared to frictional losses. The ratio is the comparison of the increase in kinetic energy relative to the flow friction loss. Analysis of the design comparison data (L) for duct lengths L 90, L 135, and L 180 show the influence of geometry on aerodynamic efficiency in the diffuser channel. Among the configurations considered, the L = 135 mm design exhibits the lowest pressure loss ($P_{loss} = 1660.02$ Pa), indicating superior performance in minimizing flow resistance. However, despite its lower pressure loss, this configuration also produces the smallest dynamic pressure difference ($\Delta P_{dynamic} = 619.43$ Pa), resulting in a kinetic energy conversion ratio of 0.37. This suggests that, although the flow is smoother, the acceleration effect due to cross-sectional changes is less pronounced compared to the other configurations.

In contrast, the L= 90 mm design yield the highest dynamic pressure difference ($\Delta P_{dynamic} = 672.19$ Pa), corresponding to the largest ratio of 0.40. This ratio indicates that the L = 90 mm design is more effective in converting energy into kinetic energy (velocity) than at reducing energy loss due to flow friction, although it also carries the highest-pressure loss ($P_{loss} = 1672.13$ Pa). The L = 180 mm configuration demonstrates intermediate performance, with a pressure loss of 1663.99 Pa and a ratio of 0.38. Technically, this difference arises from the interaction between the channel length and the diffuser angle. Sharper angles or altered flow trajectories in this L variation influence the boundary layer thickness and wall-friction intensity. Therefore, its selection should be adjusted depending on whether the objective is to minimize losses (L = 135 mm) or to maximize the kinetic energy ratio (L = 90 mm).

As shown in Figure 6, the optimization study was performed with an inlet channel bend radius (R) 90 mm and a duct brake length (L) 180 mm. Subsequently, variations in the inlet diffuser width (D) and height were examined by modifying the width and height: D 5 mm × 10 mm, D 8.75 mm × 10 mm, and D 12.5 mm × 10 mm. The D 12.5 mm x 10 mm model recorded an extremely high-

pressure loss of 15440.66 Pa but experienced an increase in dynamic pressure of 4468.26 Pa, while the D 5mm x 10mm model produced the lowest pressure loss of 4343.39 Pa and a small increase in dynamic pressure of 1550.51 Pa. In addition, the highest ratio of energy increase to loss was observed in the D 8.75mm x 10mm model at 0.36, making this the best Duct Brake dimension to apply in the second-generation Nmax vehicle.

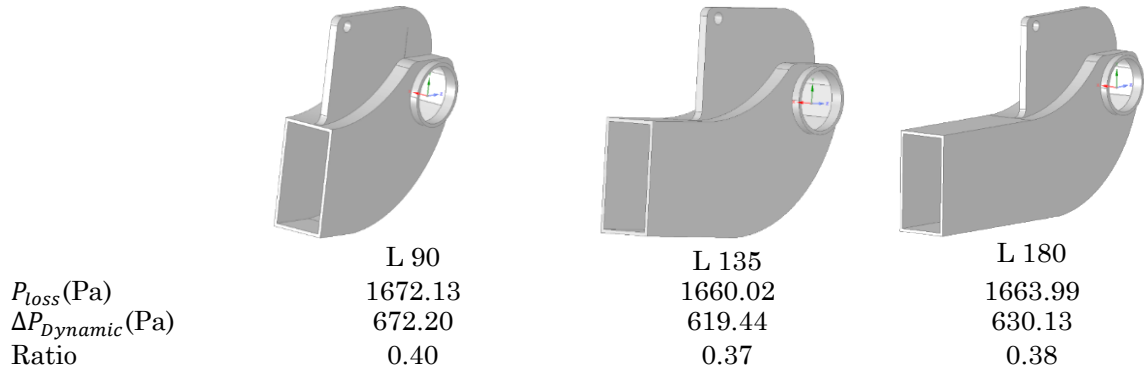


Figure 5. Flow characteristics with variations in brake duct length (L)

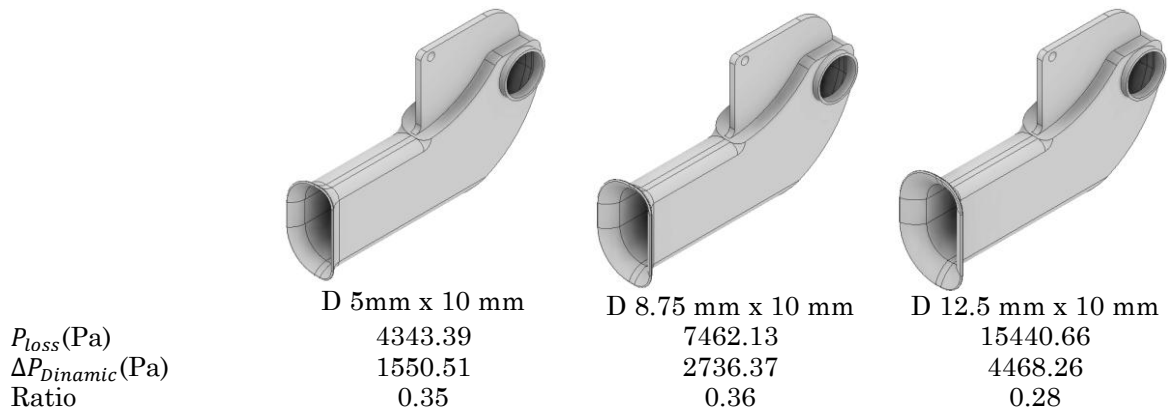


Figure 6. Characteristics of the duct brake with variations in width (D) x height of the inlet channel diffuser

CONCLUSION

This study demonstrates that CFD-based optimization of the brake duct model for the second-generation Nmax can effectively characterize the flow behavior. The optimal configuration identified in this study is defined by three principal parameters: (1) an inlet bend radius (R) of 20 mm, which can reduce the pressure loss by approximately 17.4% by mitigating turbulence; (2) a duct length (L) of 90 mm, which produces the highest kinetic energy conversion ratio of 0.40; and (3) a diffuser cross-sectional dimension (D) 8.75 mm × 10 mm, which provides the most favorable balance between dynamic pressure increase and frictional losses leading to a ratio of 0.36. Technically, this geometric combination ensures smoother, higher velocity incoming airflow while maintain controlled friction losses, thereby improving the cooling and overall efficiency. Nevertheless, experimental validation under real operating conditions on the Nmax vehicle is required to confirm the numerical findings.

REFERENCES

- [1] Gilang Satria and Azwar Ferdian, “Beda Kasus Rem Blong Truk di Indonesia dan Jepang, Bagai Bumi dan Langit” Kompas.
- [2] C. Bellini, V. Di Cocco, D. Iacoviello, and F. Iacoviello, “Temperature influence on brake pad friction coefficient modelisation,” *Materials*, vol. 17, no. 1, p. 189, 2023, doi: 10.3390/ma17010189.
- [3] M. Hongyu, “Heat dissipation type brake disc that contains rare metal,” CN111207172A, May 29, 2020

- [4] R. Jadhav, "Design and optimization of wheels for better aerodynamics and cooling of brakes," *Int J Res Appl Sci Eng Technol*, vol. 10, no. 12, pp. 418–440, 2022, doi: 10.22214/ijraset.2022.39853.
- [5] P.A. Polyakov, "Study of a segmented ventilation system of the brake disc and determination of the aerodynamic and heat exchange characteristics of the airflow," *iPolytech Journal*, vol. 25, no. 6, pp. 720–732, Jan. 2022, doi: 10.21285/1814-3520-2021-6-720-732.
- [6] M. Zhang, "Air cooling of disc brake unit by longitudinal vortex generator," 20200217378, Sep. 07, 2020
- [7] M.A. Titus, M.S. Sylvester, and A.A. Cunningham, "Active Brake Cooling Ducts," US20180313418A1, May 01, 2017 Accessed: Nov. 19, 2025. [Online]. Available: <https://patents.google.com/patent/US20180313418A1/en>
- [8] S. Singh, P. Borthakur, and S. Bahl, "A computational study on structural and thermal behaviour of disc brake rotors with varying design and material," *IOSR Journal of Mechanical and Civil Engineering*, vol. 21, no. 2, pp. 1–6, 2024.
- [9] S. Zheng, J. Ding, and J. Zuo, "Research on heat dissipation of brake disc in the semi-enclosed space under high-speed train based on fluid-solid-thermal coupling method," *Case Studies in Thermal Engineering*, vol. 56, p. 104295, 2024, doi: 10.1016/j.csite.2024.104295.
- [10] F.B. Marin and M. Marin, "CFD Modeling of Aerodynamic Car Brake Cooling System," The Annals of "Dunarea de Jos" University of Galati. *Fascicle IX, Metallurgy and Materials Science*, vol. 44, no. 4, pp. 44–47, Dec. 2021, doi: 10.35219/mms.2021.4.08.
- [11] V. K. Tripathi, R. Saini, and U.K. Joshi, "Design and analysis of ventilated disc brake by using different materials: A review," *Int J Res Appl Sci Eng Technol*, vol. 11, no. 8, pp. 627–631, 2023, doi: 10.22214/ijraset.2023.55181.
- [12] S. Shofiah, I.N. Syah, S. Hadi, N.O. Widiandaru, and L.O. Asmin, "Statistical analysis of temperature effects on brake force and efficiency in vehicle braking systems: A comprehensive study," *E3S Web of Conferences*, vol. 622, p. 01003, 2025, doi: 10.1051/e3sconf/202562201003.
- [13] A. Rashid, "Overview of disc brakes and related phenomena - A review," *International Journal of Vehicle Noise and Vibration*, vol. 10, no. 4, p. 257, 2014, doi: 10.1504/IJVNV.2014.065634.
- [14] C. Li and H.I. Yang, "Optimized shape for improved cooling of ventilated discs," *Alexandria Engineering Journal*, vol. 79, pp. 556–567, 2023, doi: 10.1016/j.aej.2023.08.035.
- [15] V.K. Agrawal *et al.*, "Optimizing ventilated disk brake design for enhanced thermal performance: an analytical and experimental approach," *Multiscale and Multidisciplinary Modeling, Experiments and Design*, vol. 8, no. 4, p. 213, 2025, doi: 10.1007/s41939-025-00797-0.
- [16] J. Dewanto, O. Soegihardjo, and A.N.R. Wijaya, "New active cooling system to prevent an overheating on the vehicle disc brake," *International Journal of Industrial Research and Applied Engineering*, vol. 3, no. 1, Apr. 2018, doi: 10.9744/jirae.3.1.1-6.
- [17] I. Feier, J. Way, and R. Redfield, "Bicycle Disc Brake Thermal Performance: Combining Dynamometer Tests, Bicycle Experiments, and Modeling," in The 13th Conference of the International Sports Engineering Association, Basel Switzerland: MDPI, Jun. 2020, p. 100. doi: 10.3390/proceedings2020049100.
- [18] M.E. Simanjuntak *et al.*, "Kinetic analysis and mathematical modeling of low-rank coal drying by using swirl fluidized bed dryer on varied angles of guide vane and temperatures," *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, vol. 46, no. 1, pp. 3263–3277, Dec. 2024, doi: 10.1080/15567036.2024.2317435.
- [19] M.E. Simanjuntak, M.B.H. Sitorus, U.J. Hutabarat, K.B.A.M. Siahaan, T. Sutrisno, and P.S. Widyawati, "Effect of mass on drying kinetics of andaliman (*Zanthoxylum acanthopodium* DC.) by swirl fluidized bed drying: A mathematical model," 2024, p. 040005. doi: 10.1063/5.0183438.
- [20] T. Sutrisno *et al.*, "Optimization of the dimple depth and arc length of dimple distance in the intake port for improved engine performance," *AIP Conference Proceedings*, p. 040004, 2024, doi: 10.1063/5.0182565.
- [21] T. Raja, G. Mathiselvan, M. Sreenivasulureddy, and X. Goldwin Xavier, "Design and analysis of Air flow duct for improving the thermal performance of disc brake rotor," *IOP Conf Ser Mater Sci Eng*, vol. 197, p. 012086, May 2017, doi: 10.1088/1757-899X/197/1/012086.