Analysis and Quantitative Effects of Rim Geometry on the Aerodynamic Performance of Production Passenger Vehicles in North America.

AP Research

04/30/2023

Word Count: 4187

Introduction

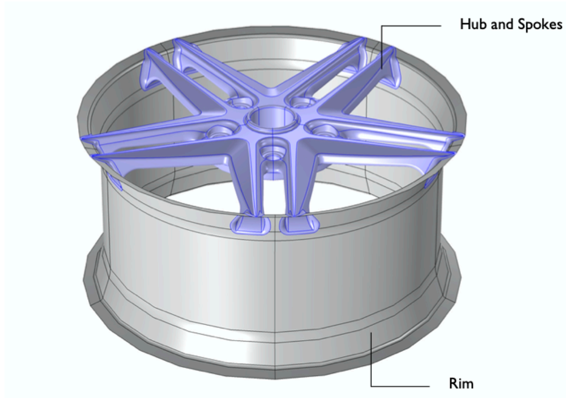
The United States Environmental Protection Agency (EPA) recently reported that transportation is responsible for an extraordinary percentage of greenhouse gas emissions within North America, specifically “over 27% of total U.S. greenhouse gas emissions” are from private vehicles (EPA, 1). This alarming statistic has caused numerous North Americans to ponder the effect that their everyday transportation choices may be having on climate change and its potential mitigation strategies.

Given these challenges, my field of research focuses on ways to leverage the aerodynamic properties of cars to reduce their carbon footprint and create environmentally-friendly transportation solutions through the analysis of numerous design components. My work can have numerous positive outcomes: improved fuel economy, enhanced vehicle performance and cost savings for consumers as well as decreased emissions.

One of the major factors affecting the aerodynamics of a vehicle is its rim geometry. Rim geometry refers to the shape, size, and angles of spokes and hubs in relation to one another (See Figure 1.).

This study's purpose is to analyze the effect of various rim designs on vehicles' aerodynamic drag coefficients[1] and identify any trends or patterns within the data. Computation Fluid Dynamics (CFD) simulations will be utilized as part of this investigation and common cars in North American markets will be targeted; results could prove invaluable in providing manufacturers and engineers with insights to increase fuel efficiency and performance of passenger vehicles.

[1] The drag coefficient, often abbreviated as "Cd", is a dimensionless number used to quantify the resistance experienced by an object when moving through a specific fluid, such as air or water.



(Figure 1.) showing the Hub and Spokes of a Rim

Literature Review

Modern advancements in computer science, computational geometry, and computational fluid dynamics (CFD) have made it possible to mitigate expensive and slow wind tunnel testing of aerodynamic performance and bring more accessibility in revolutionary ways through engineering and sciences. For instance, A study done by engineers at the University of Bucharest used CFD to reduce a vehicle's emissions levels by applying a “streamlined” body design. They applied the teardrop effect where “a liquid is placed on a flat surface and passed through the air, it will naturally form an inverse teardrop shape as the air moulds it and the forces acting on each surface reach equilibrium. Similar to an aerofoil, the teardrop is the aerodynamic ideal for an object travelling over a surface” (Ilea et all. 2018). In another study done by engineers at Chalmers University of Technology that conducted a study that used CFD with open road simulations to overpredict the drag differences between tyres with lateral and cross-sectioned groves, and use ground simulations to evaluate the interaction between the front and rear wheel tyres. Their studies use blockage correction [2](CD,corr=CD,uncorr+ΔCD,HB1+εs+εw2) to determine how differences in mechanical geometry can affect dynamic pressure, wake blockage, and flow fields (Josefsson et al. 2022).

The aerodynamic performance of a race car plays a crucial role in determining

[2] The Blockage Correction Formula is a mathematical equation designed to correct experimental data obtained during wind tunnel or fluid flow testing when there is a physical obstruction or blockage present in the test section.

its speed and handling on the track. One key aspect of the car's design that can impact its aerodynamics is the geometry of the rims. Analyse factors such as drag, lift, and turbulence. A study done by engineers at the international sports engineering association studied the direct and indirect cause-and-effect relationships between the F1 Cars' aerodynamic substances of open wheel and rim designs. The authors believe that analyzing the flow field of an F1 Car can result in the innovation of a new F1 aerodynamic subsystem technology. These authors use computational fluid dynamics, and smoke wind tunnel testing of a 40% scale F1 car to provide data for the flow fields and aerodynamic drag of an F1 car. Additionally, the authors provide explanations for how the computational fluid dynamic visualization resulted in the improvement of frontal wing technology. The author's conclusion interests me in my research on how different subsystems of a car can greatly influence the aerodynamic properties of a separate part (Keller et al. 1999).

The current abundance of research on improving efficiency around gasoline combustion engines by researching exhaust emissions and engine efficiency has prompted others to research the indirect correlations of efficiency in production vehicles. In a study done by engineers at Honam University, they examine how the concept of an “air jet wheel deflector can reduce vehicle aerodynamic drag” through an experimental studying using CFD (Lee et al. 2018). Their findings through parametric studies revealed that an “air jet wheel deflector system effectively reduces vehicle aerodynamic drag by up to 6.4%” (Lee et al. 2018). Another study done by engineers at Honam University found that the position of the wheel wells in relation to the rest of the vehicle can affect the aerodynamic performance. The study found that moving the wheel wells closer to the center of the vehicle can reduce drag and improve aerodynamic performance (Lee et al. 2015). In Omer Faruck Cavusoglu peer reviewed masters thesis from the Department of applied mechanics at the Chalmers University of Technology. Cavusoglu's study explored the field of aerodynamics around the wheel housing of a car by analyzing the fluid dynamics around drag, pressure, turbulence and the correlation between rolling resistance and aerodynamic drag to advance the notion of electric car technology. Cavusoglu arrived at the conclusion that the lower half of a car (underbody, wheels, etc) accounts for over 30% of the aerodynamic drag in an average sedan-bodied car (Cavusoglu 2017). In another article by engineers at Rice University concentrates on the computational fluid dynamics of F1 car wheel tire deformation under different rotational speeds and aerodynamic loads. The engineers believe that the isogeometric analysis framework for cars and tire aerodynamics can determine road contact and tire deformation and rotation. These authors use isogeometric analysis to provide evidence that "the geometries of the computational models for the car body and tires are close to the actual geometries" (Kurashi et al. 2022). Additionally, the authors provide evidence for "the effectiveness of the analysis framework they have built, including generating NURBS meshes for car and tire models with complex, realistic geometries." (Kurashi et al. 2022).

Despite the importance of rim geometry in determining a vehicle's aerodynamic performance, there has been limited research on the specific effects of rim geometry on this performance. For instance, Cavusoglu's study determines the aerodynamic effects of wheel housing by creating original models using selected data sets to help improve aerodynamic performance to help improve fuel economy and efficiency of gasoline combustion engines. Cavusoglus research addresses the gap in determining indirect causes of the efficiency of production passenger vehicles but lacks the focus on the correlation between innovating rim technology. Additionally, the majority of previous research has focused on the aerodynamics of sports vehicles, rather than the production of passenger vehicles that make up the majority of the North American vehicle market. For instance, Keller's research on the direct and indirect cause-and-effect relationships between the F1 car's aerodynamic substances of open wheel and rim design reveals the gap in research where research on passenger production cars' aerodynamic performance is considered less compared to performance cars. This gap in the literature presents an opportunity for further study on the effects of rim geometry on the aerodynamic performance of production passenger vehicles in North America. This research paper aims to address this gap by analyzing the quantitative effects of various rim geometries on the aerodynamic performance of production passenger vehicles in North America and to provide valuable insights for automotive engineers and designers to improve the aerodynamic performance of these vehicles.

Methodology

CFD simulations will be used to analyze the correlation between rim geometry and aerodynamic performance of production passenger vehicles in North America. A 3-D scaled model of the rim will be used as the input for the CFD simulations.

Taking into consideration of CAD resources we will be taking the 3 most popular production passenger vehicle styles; Truck; SUV, and Sedan. According to Joey Cappaerla at Car and Driver, the most-sold production passenger vehicles in North America in 2022 are the Ford F150 as the most-sold commercial pickup truck; The Toyota Rav 4 as the most-sold SUV; and The Toyota Camry as the most-sold sedan. The selection of rims of these vehicles will be determined by the most popular vehicle trim of each model for analysis. A standard steel wheel will be used for comparison.

CAD

CAD stands for Computer-Aided Design and refers to using computer software to design 2D or 3D models of physical objects for purposes of modification, analysis, and optimization.

CAD software is widely utilized across disciplines such as engineering, architecture and industrial design to produce precise technical drawings and models. Furthermore, its virtual environment enables designers to visualize and test their designs to identify any potential problems before final production or construction begins.

CAD software typically features tools for creating and altering geometries such as lines, curves, surfaces and solids. These are often supplemented by special features like parametric modelling which enables designers to define design parameters easily before creating designs that can be changed according to changes. Other tools may include tools for analyzing the structural integrity of designs as well as simulating behaviour under various conditions as well as creating photorealistic renditions.

CAD has revolutionized the way designs are created and produced, enabling designers to work more quickly and with greater precision. Collaboration on designs has also become easier as files can easily be shared between multiple users for editing purposes. As such, CAD has become an essential tool across industries in speeding up design processes while decreasing costs.

AutoCAD will be used to measure the geometric variables of the rims. Variables such as the Rim diameter (the distance across the circular inside edge of the rim, measured in meters .) Rim offset (the distance between the centerline of the rim and the hub mounting surface, measured in meters), Surface area (how much area the spokes and hubs occupy within the surface of the rim). This method can provide a detailed representation of the rim shape and curvatures.

CFD

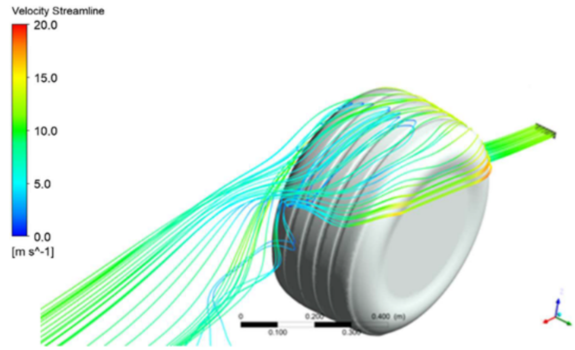
CFD stands for Computational Fluid Dynamics and refers to an area of fluid mechanics which uses numerical analysis and algorithms to study fluid flows (liquids or gases) as well as heat transfer issues.

CFD involves discretizing and solving numerically the governing equations of fluid dynamics using powerful computer algorithms in order to simulate physical phenomena like turbulence, heat transfer, combustion, and multiphase flows.

CFD can be applied across many disciplines, including aerospace engineering, automotive engineering, chemical engineering, and environmental engineering. CFD simulations are frequently employed to optimize engineering systems like aircraft engines, heat exchangers and chemical reactors as well as predict their behaviour in natural systems like ocean currents and atmospheric flows.

CFD has revolutionized how engineers and scientists study fluid dynamics, providing more accurate predictions with faster design iteration times and simulating fluid phenomena that would otherwise be difficult or impossible to study experimentally. CFD has become an essential tool in numerous industries for optimizing the design and performance of complex systems and processes.

CFD simulations will be used to analyze the aerodynamic performance of the selected rims. A 3-D scaled model of each rim will be used as the input for the CFD simulations using the software SimScale. The CFD simulations will provide a detailed analysis of the flow field around the rim, including the velocity and pressure fields, as well as the drag and lift coefficients (Refer to Figure 2.).



(Figure 2.) CFD of a solid wheel rim with tires showing the flowfield.

Mesh

CFD utilizes meshes as discrete representations[3] of continuous domains[4] to numerically solve fluid flow and heat transfer equations.

Meshes in CFD are collections of small interconnected cells that form an abstraction layer across the computational domain. Each cell possesses its own specific set of properties such as size, shape, orientation and geometry that define them from other cells.

---

[3] Discrete representation refers to a type of data representation that deals with discrete, separate, and discontinuous values.
[4] Continuous domain is defined as any set of values which spans an extended range or interval.

CFD utilizes several types of meshes, including structured, unstructured and hybrid meshes. A structured mesh consists of regular rows of cells in an organized grid pattern; unstructured mesh cells vary widely in shape and size depending on their placement; hybrid meshes combine elements from each style (see Figure 3.).

Mesh quality is a critical factor when conducting CFD simulations, with accurate and reliable results coming from using high-quality meshes while low-quality ones may produce numerical errors and inaccuracies. Mesh density also plays an important role, with higher densities yielding more precise simulation results but increasing computational times accordingly.

CFD simulations often rely on special software tools that automatically create meshes based on the geometry of the computational domain and desired levels of accuracy, while selecting mesh types and densities depending on both specific issues being solved and available computational resources.

SimScale will be used to compute meshes using the Hex Dominant algorithm generating a mesh that consists of 6-9 million cells per computation and CAD model.

Algorithm

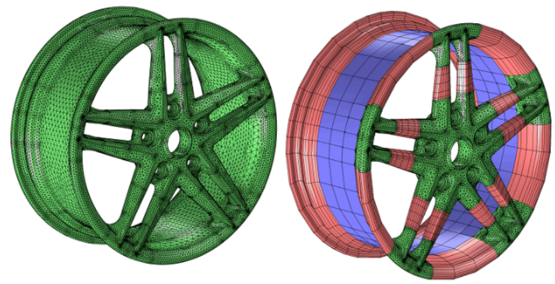
The Hex dominant algorithm is a meshing technique used in computational fluid dynamics to generate high-quality hexahedral meshes for complex geometries. Hexahedral meshes[5] are desirable because they are computationally efficient and provide accurate solutions to fluid flow problems.

[5] Hexahedral meshes, are three-dimensional mesh structures used for three-dimensional modelling applications.

When using the Hex dominant algorithm, the focus is on creating a coarse mesh that captures the overall geometry of the object being analyzed, rather than the surface texture. This is achieved by creating larger hexahedral elements that span the entire thickness of the object, rather than using smaller elements that conform closely to the surface.

By creating larger elements, the Hex dominant algorithm is able to effectively isolate the computational domain[6] (CD) to the interior of the object being analyzed, while excluding the surrounding fluid domain and the surface texture (Refer to Figure 3.). This is because the larger elements are able to accurately represent the geometry of the object, without requiring the use of smaller elements that would capture the surface texture.

[6] A computational domain, refers to the region or space, in which a numerical simulation or computational analysis is performed.



(Figure 3.) On the Left, the model shows an unstructured mesh, On the right a hybrid of structured mesh (Pink and Purple) with unstructured mesh (green). Green Rim is generated with a Hex Dominant Algorithm.

Analysis

CFD simulation results for various variations of rim geometry will be compared and evaluated to ascertain their effect on the aerodynamic performance of each vehicle, specifically with regard to drag coefficient.

One way of visualizing results is with a graph, in which the x-axis represents different drag coefficients while the y-axis indicates aerodynamic performance parameters. Each point on this graph represents the results from CFD simulations for each rim model.

By comparing results, it will be possible to identify trends and patterns within the data and draw conclusions about how rim geometry affects the aerodynamic performance of each vehicle. For instance, an increase in drag coefficient with increasing width might suggest wider rims have an advantageous effect in terms of decreasing drag.

By comparing simulation results across vehicle models, the study can identify whether there are any variances in how rim geometry impacts aerodynamic performance between them.

Validation will take place through an original 3-D model which employs positive correlational characteristics between rim geometry and aerodynamic performance.

Results

Variable	**2022 Ford F150**	**2022 Toyota Camry**	**2022 Toyota Rav 4**	**Base model Steelie**
Rim Diameter	0.51 m	0.46 m	0.43 m	0.41 m
Rim Hub Off-set	0.044 m	0.035 m	0.039 m	0.020 m
Rim Surface Area	0.41 m^2	0.34 m^2	0.35 m^2	0.53 m^2

Cd vs Rim (Ford F150) [20”]

Cd	**m/s**
0.211	**5**
0.226	**10**
0.228	**15**
0.324	**25**

Cd vs Rim (Toyota Rav 4) [17”]

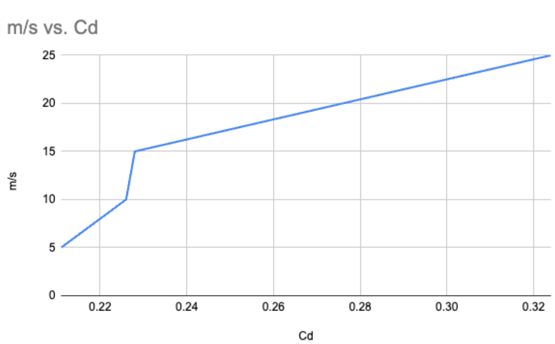
Cd	**m/s**
0.192	**5**
0.198	**10**
0.205	**15**
0.250	**20**

Cd vs Rim (Toyota Camry) [18”]

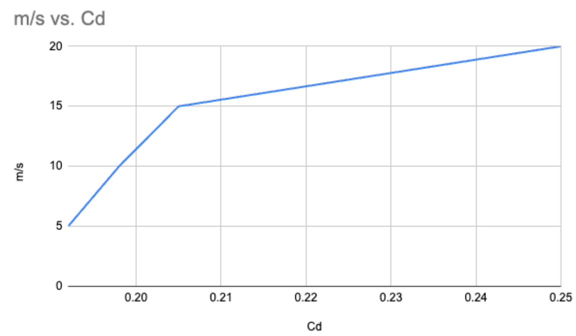
Cd	m/s
0.199	5
0.202	10
0.209	15
0.289	25

Cd vs Rim (Steelie) [16”]

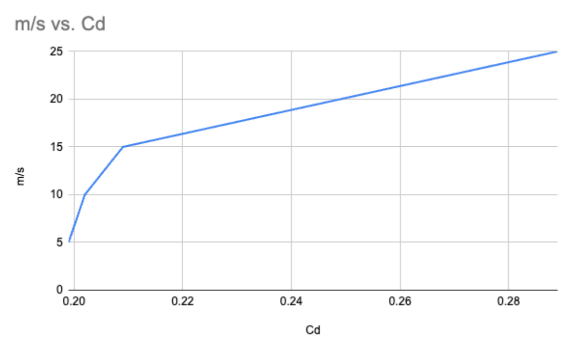
Cd	m/s
0.221	5
0.236	10
0.239	15
0.312	25



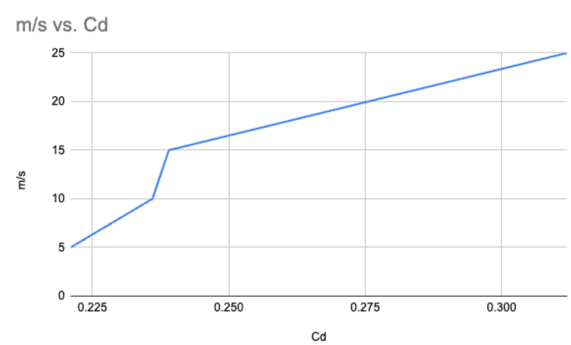
Ford F150: Average Cd = 0.247 at average speed = 14 m/s (50 km/h)



Toyota Rav4: Average Cd = 0.211 at average speed = 14 m/s (50 km/h)



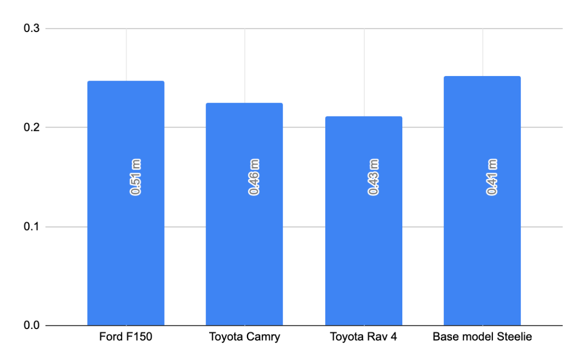
Toyota Camry: Average Cd = 0.225 at average speed = 14 m/s (50 km/h)



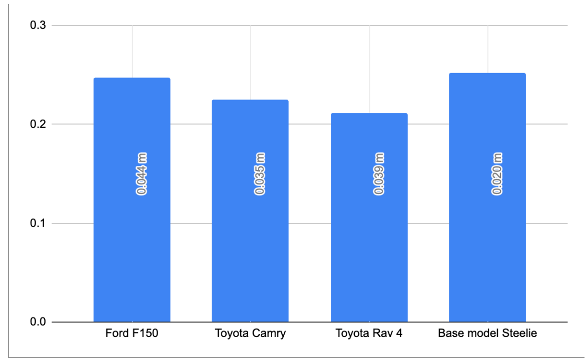
Standard Steel Rim: Average Cd = 0.252 at average speed = 14 m/s (50 km/h)

Correlation Analysis

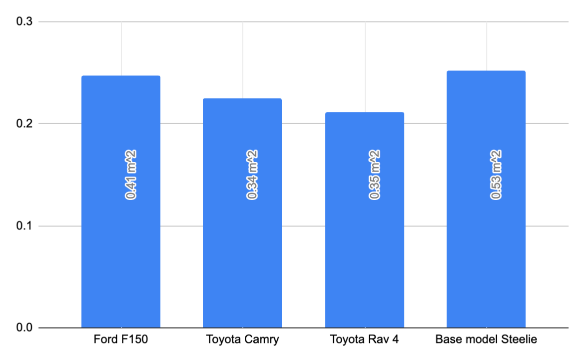
Variable	Rim Diameter	Cd
Ford F150	0.51 m	0.247
Toyota Camry	0.46 m	0.225
Toyota Rav 4	0.43 m	0.211
Base model Steelie	0.41 m	0.252



Variable	Rim Diameter	Rim Hub Off-set
Ford F150	0.51 m	0.044 m
Toyota Camry	0.46 m	0.035 m
Toyota Rav 4	0.43 m	0.039 m
Base model Steelie	0.41 m	0.020 m



Variable	Rim Surface Area	Cd
Ford F150	0.41 m^2	0.247
Toyota Camry	0.34 m^2	0.225
Toyota Rav 4	0.35 m^2	0.211
Base model Steelie	0.53 m^2	0.252



Discussion

Researching the effect of rim geometry on the aerodynamic performance of production passenger vehicles is an integral component of modern vehicle research, as this can have serious ramifications for vehicle fuel economy, emissions and overall performance. For this correlation study, I examined four North American passenger vehicles: 2022 Ford F150 with 20" rims; 2022 Toyota Camry with 18" rims; 2022 Toyota Rav 4 with 17" rims and the base rim for all car models: 16" steel rims.

Rim Diameter

Its Rim diameter is an integral component of aerodynamic performance for vehicles. When increasing rim diameter, tire surface area increases which leads to greater drag. Based on available data, Ford F150 boasts the largest rim diameter with 0.51 meters followed by Toyota Camry (0.46 meters), Toyota Rav 4 (0.43 meters), and Steelie Base Model (0.41 meters).

Rim Hub Offset

Rim Hub offset is the distance between the hub mounting surface and the centerline of a wheel, which can alter and affect the aerodynamic performance and properties of cars. A larger hub offset usually results in wider track width, which increases stability but may increase drag. Based on data collected by Autobytel.org, Ford F150 boasts the largest hub offset at 0.044 meters followed by the Toyota Camry (0.035), Rav 4 (0.039) and base model Steelie with respective offsets of 0.035, 0.039, and 0.020 metres respectively.

Rim Surface Area

Rim surface area, or the total area of a wheel's surface, can have a significant effect on the aerodynamic performance of vehicles. As more surface area increases drag force is exerted upon it. According to the data provided here, the base model Steelie had the largest surface area at 0.53 m2, followed by Ford F150, Toyota Rav 4 and Camry which each have 0.41m2, 0.35m2 and 0.34 m2, respectively.

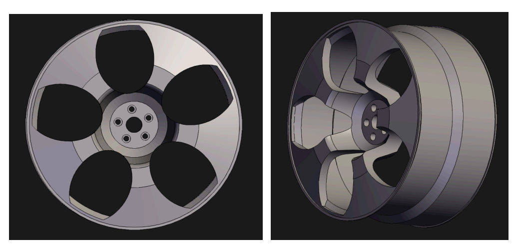
Analysis

My analysis shows that vehicle characteristics such as rim diameter, hub offset, and surface area have an important effect on its aerodynamic performance. In general, larger rim diameters lead to worse aerodynamic performance as indicated by higher Cd values.

Similarly, hub offset can also impact aerodynamic performance, with greater offset values generally leading to worse performance. However, the effect of the hub offset appears to be less significant than that of the rim diameter.

Finally, my results indicate that rim surface area can also impact aerodynamic performance, with larger surface areas generally leading to worse performance. This is likely due to the fact that larger rim surface areas increase the overall drag coefficient of the vehicle.

In consideration of errors, applying my findings is crucial to validate my results. Having an understanding where smaller rims, smaller hub offset, and lower surface area, an original rim was designed in Blender (Refer to Figure 4.).



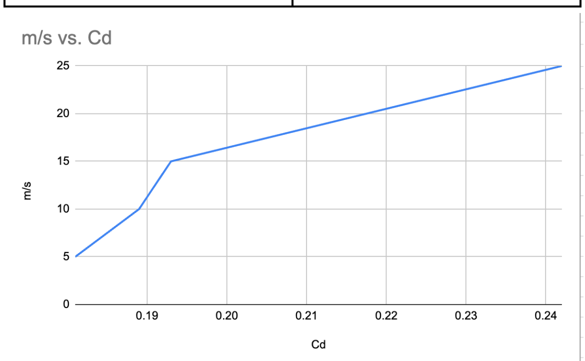
(Figure 4.) To the left a top view of the original rim. To the right an angled view of the rim.

Referring to Figure 4 the original design depicts a flatter surface area with ovals slotted out to create a five-spoke design. The slots are not only meant to decrease surface area but to ventilate heat when braking. It is important to take into consideration of external factors, not just focus on aerodynamics when applying these results for functional purposes.

Variable	**Original Design Rim**
Rim Diameter	0.41m
Rim Hub Off-set	0.00m
Rim Surface Area	0.25m

Cd vs Original Rim [16”]

Cd	**m/s**
0.181	**5**
0.189	**10**
0.193	**15**
0.242	**25**



Original Design: Average Cd = 0.201 at average speed = 14 m/s (50 km/h)

My study on the aerodynamic performance of passenger vehicle rims has yielded valuable insights into the impact of rim geometry on vehicle performance. By analyzing the data for different rim sizes, hub offsets, and surface areas, we have identified the key factors that affect the aerodynamic drag coefficient of passenger vehicles.

The findings of my study have been used to design an original rim using AutoCAD, which features a flatter surface area with ovals slotted out to create a five-spoke design. The slots not only decrease surface area but also provide ventilation to dissipate heat when braking. It is important to consider external factors, such as braking performance, in addition to aerodynamics when designing a functional rim.

My results show that the original rim design has a significantly lower Cd than the base model steelie (16" rim) at various speeds. This confirms that the design principles we have employed can be effective in improving the aerodynamic performance of passenger vehicles. However, it is important to note that there may be errors in my measurements and modelling, and further testing is necessary to validate the effectiveness of my design in real-world driving conditions.

Limitations

While my study offers many insights into how rim geometry impacts passenger vehicles' aerodynamic performance, there are certain constraints which should be kept in mind when reading our findings.

My research focused solely on four specific vehicle models, which may not be reflective of all passenger vehicles in North America. Thus, the findings from my study must be generalized with caution.

My study did not take into account the effects of tire design and inflation pressure, both of which can greatly alter the aerodynamic performance of vehicles. Therefore, future studies must take these elements into account as well.

This research highlights the significance of rim geometry, it does not explore all factors contributing to vehicle aerodynamics. Other design elements, including body shape, underbody design and additional vehicle features can significantly influence aerodynamic performance; future studies should aim to incorporate them all into an in-depth analysis of vehicle aerodynamics.

My analysis was performed through computational simulations, which may not accurately reflect real-world driving conditions. Therefore, additional vehicle tests are necessary in order to validate my findings and optimize rim design for maximum aerodynamic performance.

Additionally, this study assumes a uniform driving condition without considering factors like wind speed, road gradient or differences in driving style - all factors which could potentially have an impactful aerodynamic performance of vehicles and be overlooked in such studies. Their exclusion may restrict the generalisability of findings.

While my study provided a solid base for future research on rim geometry and its influence on vehicle aerodynamics, more research must be conducted in order to address its limitations as well as fully grasp all interactions among vehicle components that impact aerodynamic performance.

Conclusion and Implications

The automotive sector may benefit greatly from my study, particularly when it comes to creating more aerodynamic and fuel-efficient vehicles.

The importance of rim shape for vehicle aerodynamic performance is examined in my research. It provides quantifiable information on how various designs affect drag coefficients and other aerodynamic parameters of production passenger cars. Automotive manufacturers could employ this study to optimize designs to increase both performance and fuel efficiency.

The results of this research can be used to create new standards and rules for vehicle aerodynamics. It could serve as guidelines for engineers and other parts that increase aerodynamic drag for governments and regulatory bodies. This would result in more consistent industry-wide standards and more environmentally friendly, fuel-efficient vehicles.

The potential for further optimization of vehicle components beyond traditional aerodynamic surfaces such as the body and underbody. By considering the impact of rim geometry on aerodynamic performance, designers can explore new ways to reduce drag and improve efficiency in vehicles.

My research emphasizes how crucial it is to carefully take into account rim geometry when developing rim models for mass manufacturing. There is tremendous potential for cross-disciplinary research collaborations. The findings from this study can spur collaborations among automotive engineers, material scientists, computational modellers, sustainability and renewable energy experts, manufacturing techniques experts to explore innovative materials and manufacturing processes that further increase aerodynamic performance. Such efforts may also foster vehicle designs which not only optimize aerodynamic performance but are also environmentally sustainable.

This study's ramifications extend far beyond manufacturing. The insights gained can be used to inform and educate consumers on the significance of aerodynamic performance and fuel efficiency when making vehicle selection decisions. By raising awareness and encouraging more fuel-efficient designs to be adopted more widely, collectively we can contribute towards reduced carbon emissions and decrease our environmental footprint through transportation.

Works Cited

Josefsson, E., Hobeika, T., & Sebben, S. (2022). Evaluation of wind tunnel interference on numerical prediction of wheel aerodynamics. *Journal of Wind Engineering & Industrial Aerodynamics*, *224*, N.PAG. https://doi.org/10.1016/j.jweia.2022.104945

Lee, S. W. (2018). Computational analysis of air jet wheel deflector for aerodynamic drag reduction of road vehicle. *Microsystem Technologies*, *24*(11), 4453–4463. https://doi.org/10.1007/s00542-018-3992-1

Spagnolo, S., Zhang, X., Hu, Z., Stalnov, O., & Angland, D. (2017). Unsteady aerodynamics of single and tandem wheels. *Journal of Fluids & Structures*, *69*, 121–136. https://doi.org/10.1016/j.jfluidstructs.2016.11.022

. A. S, D. B. A. S., as., J., & . R, C. (2018). *Design and analysis of alloy wheel for multi-purpose vehicle*. Research Gate. Retrieved November 8, 2022, from https://www.researchgate.net/publication/328364931_DESIGN_AND_ANALYSIS_OF_ALLOY_WHEEL_FOR_MULTI-PURPOSE_VEHICLE

CAVUSOGLU, Ö. M. E. R. F. A. R. U. K. (2017). *Chalmers Publication Library (CPL): Förstasida*.CHALMERS UNIVERSITY OF TECHNOLOGY. Retrieved November 7, 2022, from https://publications.lib.chalmers.se/records/fulltext/250512/250512.pdf

Buck, P. de, & A Martins, J. R. R. (2020). *Minimum lap time trajectory optimisation of performance vehicles with four-wheel drive and active aerodynamic control*. Taylor & Francis. Retrieved November 7, 2022, from https://www.tandfonline.com/doi/abs/10.1080/00423114.2022.2101930?journalCode=nvsd20

Kuraishi, T., Takizawa, K., & Tezduyar, T. (2022). *Space–time isogeometric analysis of car and tire aerodynamics with road ...* Springer. Retrieved November 8, 2022, from https://www.researchgate.net/publication/359678287_Space-time_isogeometric_analysis_of_car_and_tire_aerodynamics_with_road_contact_and_tire_deformation_and_rotation

Kellar, Pearse, & Savill. (1999). Formula 1 car wheel aerodynamics. *Sports Engineering (International Sports Engineering Association)*, *2*(4). https://doi.org/10.1046/j.1460-2687.1999.00030.x

Teutsch, C., & Berndt, D. (2006, October). *Automated geometry measurement of wheel rims based on optical 3D metrology*. Research Gate. Retrieved November 7, 2022, from https://www.researchgate.net/publication/228933141_Automated_geometry_measurement_of_wheel_rims_based_on_optical_3D_metrology

Huminic, A., & Huminic, G. (2017, April 1). *Aerodynamic study of a generic car model with wheels and underbody diffuser - International Journal of Automotive Technology*. SpringerLink. Retrieved November 7, 2022, from https://link.springer.com/article/10.1007/s12239-017-0040-6