BRNO UNIVERSITY OF TECHNOLOGY

VYSOKÉ UČENÍ TECHNICKÉ V BRNĚ

FACULTY OF MECHANICAL ENGINEERING

FAKULTA STROJNÍHO INŽENÝRSTVÍ

ENERGY INSTITUTE

ENERGETICKÝ ÚSTAV

RACE CAR BRAKE COOLING DEVELOPMENT

NÁVRH CHLAZENÍ BRZD ZÁVODNÍHO VOZU

BACHELOR'S THESIS

BAKALÁŘSKÁ PRÁCE

AUTHOR — Štěpán Joura

AUTOR PRÁCE

SUPERVISOR — Ing. Pavel Novosad

VEDOUCÍ PRÁCE

BRNO 2025

Assignment Bachelor's Thesis

Institut:	Energy Institute
Student:	**Štěpán Joura**
Degree programm:	Fundamentals of Mechanical Engineering
Branch:	Fundamentals of Mechanical Engineering
Supervisor:	**Ing. Pavel Novosad**
Academic year:	2024/25

As provided for by the Act No. 111/98 Coll. on higher education institutions and the BUT Study and Examination Regulations, the director of the Institute hereby assigns the following topic of Bachelor's Thesis:

Race car brake cooling development

Brief Description:

Developing an efficient brake cooling duct for the Formula Student car is essential to ensure optimum performance and safety during racing. The intense braking required in competition generates a significant amount of heat, which if not managed correctly can lead to reduced braking efficiency and brake fade. By designing a brake cooling system, brake temperatures can be reduced, improving braking performance, extending the life of brake components and improving driver control. The design of these components represents a compromise between reliability and vehicle performance. It is therefore advisable to consider their optimal design. At the same time, the manufacturability of the designed components must be taken into account.

Bachelor's Thesis goals:

Design brake cooling air duct geometry for Formula Student car

Perform thermal flow simulation

Optimise the position of the outlet in relation to the brake caliper

Recommended bibliography:

Versteeg, H. and Malalasekera, W. An Introduction to Computational Fluid

Dynamics: The Finite Volume Method. Pearson Education Limited, 2007. ISBN 978-0-1312-7498-3.

MILLIKEN, Douglas L. Race car vehicle dynamics problems, answers and experiments. Warrendale: SAE International, 2003. ISBN 07-680-1127-2.

LIMBERT, Rudolf. Brake design and safety. 3rd ed. Warrendale: SAE

International, 2011, 432 s. ISBN 978-0-7680-3438-7.

Deadline for submission Bachelor's Thesis is given by the Schedule of the Academic year 2024/25

In Brno,

L. S.

prof. Ing. Jiří Pospíšil, Ph.D.
Director of the Institute

doc. Ing. Jiří Hlinka, Ph.D.
FME dean

ABSTRACT

The primary objective of this bachelor's thesis is the development of a brake cooling system for the ED5 Formula Student monopost from team TU Brno Racing. Based on research and the spatial constraints imposed by the vehicle's design, the implementation of brake ducts was identified as the most suitable brake cooling solution due to their simplicity, effectiveness, and comparatively low weight. Three distinct configurations of brake ducts were proposed, each differing in the outlet position relative to the brake caliper, with the aim of achieving maximum cooling performance. To evaluate the effectiveness of these configurations, computational fluid dynamics (CFD) simulations were conducted at a reference vehicle speed of 60 km/h, incorporating average heat flux values obtained from telemetry data of previous TU Brno Racing vehicles. The results of the simulations served as the basis for selecting the most efficient design.

Key words

Disc brakes, brake duct, heat dissipation, computational fluid dynamics (CFD)

ABSTRAKT

Hlavním cílem této bakalářské práce je návrh chladicího systému brzd pro monopost Formule Student ED5 týmu TU Brno Racing. Na základě provedené rešerše a prostorových omezení daných konstrukcí vozidla bylo jako nejvhodnější řešení chlazení brzd určeno použití chladících kanálů, a to díky jejich jednoduchosti, účinnosti a relativně nízké hmotnosti. Byly navrženy tři odlišné konfigurace chladících kanálů, které se liší polohou výstupu vzhledem k brzdovému třmenu, s cílem dosáhnout maximálního chladicího výkonu. Pro zhodnocení účinnosti těchto konfigurací byly provedeny simulace pomocí computational fluid dynamics (CFD) při referenční rychlosti vozidla 60 km/h, přičemž byly zohledněny průměrné hodnoty tepelného toku získané z telemetrických dat předchozích vozidel týmu TU Brno Racing. Výsledky těchto simulací posloužily jako podklad pro výběr nejefektivnějšího návrhu.

Klíčová slova

Kotoučové brzdy, chladící kanál, odvod tepla, computational fluid dynamics (CFD)

ROZŠÍŘENÝ ABSTRAKT

Hlavním cílem této bakalářské práce je návrh, simulace a implementace systému chlazení brzdového mechanismu monopostu Formule Student pro tým TU Brno Racing. Práce se zaměřuje na optimalizaci odvodu tepla z brzd, aby byla zajištěna jejich spolehlivost a výkonnost.

Vůz pro sezónu 2024/2025, s označením ED5, představuje pro tým významný technologický pokrok. Klíčovou změnou je přechod od pohonu dvou kol k pohonu všech čtyř kol, což přináší nové technické výzvy, ale zároveň otevírá prostor pro další inovace a vývoj. Přehřívání předních brzd, jež představovalo problematický aspekt již u předchozích konstrukčních řešení, se stalo kritickým faktorem vzhledem k integraci elektrických motorů přímo do nábojů kol. Tato konstrukční změna navíc znemožnila využití dříve používaného systému chlazení, založeného na směrových kanálech (náfucích), které přiváděly vzduch k brzdovému kotouči a třmenu.

Rešeršní část této bakalářské práce se věnuje analýze brzdového systému implementovaného v monopostu ED5. Zaměřuje se na klasifikaci různých typů brzd, jejich konstrukčních řešení, variant a metod chlazení. Pro každý typ a konstrukční přístup je uvedeno zdůvodnění jeho vhodnosti či nevhodnosti pro dané použití. Závěr této části se zabývá rovněž problematikou uchycení chladicího kanálu k elektrickému motoru vozu.

Následující části práce jsou věnovány přípravě a provedení simulací pomocí computational fluid dynamics (CFD). Popis postupu přípravy simulace začíná charakteristikou geometrie použité pro simulaci společně s vysvětlením jejího zjednodušení. Dále jsou představeny tři odlišné konstrukční návrhy chladicích kanálů a geometrie kotoučové brzdy. Závěrem je uvedena varianta simulace bez přítomnosti kanálu, jejímž cílem je zhodnocení a kvantifikace významu chladicího systému.

Po představení výpočtové domény jsou detailně popsány povrchová a objemová síť, společně s nastavením zahrnujícím zjemnění zakřivení a hran u povrchové sítě a implementaci prismatických vrstev u objemové sítě, jejichž správná konfigurace zásadně ovlivňuje přesnost výsledků.

Definice výpočtového modelu zahrnuje charakteristiku nastavení modelování turbulence, včetně představení použití metody Reynolds-Averaged Navier-Stokes (RANS) v kombinaci s modelem k-omega SST. Dále je zdůrazněna důležitost zahrnutí rovnice energie do výpočtů pro simulaci odvodu tepla z brzd. Pro přesné zachycení tohoto jevu je klíčové také vhodné definování materiálů a jejich vlastností. V rámci této simulace byly implementovány tři odlišné okrajové podmínky pro modelování rotace jednotlivých částí geometrie, přičemž každá vyžadovala specifický přístup k zajištění věrného napodobení reálných podmínek.

Nastavení řešiče zahrnovalo implementaci prostorové diskretizace druhého řádu za účelem dosažení vyšší přesnosti výsledků a aplikaci pseudo time metody umožňující zrychlení výpočtů díky odlišnému nastavení časového kroku pro pevné látky a kapaliny. Závěrečným krokem před spuštěním simulace byla inicializace řešiče, k níž byla využita hybridní metoda inicializace.

Výsledky CFD simulací prokázaly klíčový význam použití chladícího kanálu, přičemž všechny navržené konstrukce umožnily snížit povrchovou teplotu třmene až o 220 Kelvinů a průměrnou teplotu brzdového kotouče o přibližně 200 Kelvinů. Navíc byla potvrzena správnost nastavení sítě na základě veličiny y^+ a celková validita použitého nastavení simulace prostřednictvím vizuální analýzy odvodu tepla z kotouče během jeho rotace.

Ze tří návrhů chladicího kanálu byl vybrán a později použit na monopostu ED5 ten, který vykázal nejvyšší chladicí efekt třmene brzdy. Tato volba byla provedena na základě obav z vysokých teplot této komponenty v simulacích, spojených s rizikem překročení teploty varu hydraulické kapaliny, což by mohlo způsobit selhání brzdového systému. Výsledky simulací naznačující překročení této teploty jsou však zavádějící, protože nastavení simulace vychází z předpokladu extrémních podmínek, které se ani v náročných závodních podmínkách Formule Student takřka nevyskytují. I přesto bylo doporučeno pokračovat v optimalizaci této specifické konstrukce chladicích kanálů v kombinaci s implementací motorového brzdění.

BIBLIOGRAPHIC CITATION

JOURA, Štěpán. *Race car brake cooling development*. Online, bachelor's Thesis. Pavel NOVOSAD (supervisor). Brno: Brno University of Technology, Faculty of Mechanical Engineering, 2025. Available at: https://www.vut.cz/en/students/final-thesis/detail/168719. [accessed 2025-05-21].

DECLARATION

I hereby declare that I have prepared my bachelor thesis on Race car brake cooling development independently using the literature and sources listed at the conclusion of this thesis.

In Brno, on 23 May 2025 …………………………………..
Štěpán Joura

ACKNOWLEDGEMENTS

I would like to express my deepest gratitude to my supervisor Ing. Pavel Novosad for his generous assistance and valuable advice. I would also like to thank the members of TU Brno Racing for the opportunity and the amount of knowledge they provided throughout the creation of this thesis. Additionally, I would like to express my gratitude to Ing. Jiří Vondál, Ph.D for providing valuable insight into the issues of CFD simulations crucial to this thesis.

CONTENTS

INTRODUCTION

The fact that the braking system is a critical component in ensuring the safe operation of any vehicle is undeniable. Therefore, there can be no dispute regarding the reliability of the braking system. In road cars, the occurrence of brake failure due to overheating is quite rare. However, in race cars and monoposts, this problem becomes a significant concern.

Nowadays the norm, not only for high-performance applications, is the predominant utilization of air-cooled friction disc brakes. This approach offers the best performance not only in terms of cooling effectiveness but also in terms of repairability and maintenance. A substantial body of research has been dedicated to the optimization of brake discs and calipers with the objective of delaying overheating. Solutions such as drilled and grooved surfaces of discs for enhanced heat exchange with the surrounding air flow and inner cooling, in combination with ducts directing air into and through the disc, along with material innovations, have emerged.

In the professional racing series, an abundant use of carbon brakes can be seen, which are regarded as the optimal solution for high-demand applications. Carbon brakes demonstrate resilience against high temperatures, rapid heat loss, and a lower weight when compared to alternative options. However, they lack performance at lower speeds and temperatures, to the extent that they can be dangerous when operating outside of the ideal conditions. Therefore, alternative solutions need to be designed, addressing the specific needs of each application.

The focus of this bachelor thesis is to design, simulate and implement a cooling solution for the brake system of the Formula Student monopost, for the TU Brno Racing team. The transition from the two-wheel drive to the four-wheel drive necessitated a redesign, with the objective of maintaining the vehicle's low weight and dealing with the constraints imposed by the suspension system and new wheel hub design.

1 FORMULA STUDENT

Formula Student is a competition in which university-based teams from around the world compete in races composed of numerous separate challenges. These challenges test various aspects of the race car and the team behind it, ranging from the vehicle characteristics, such as acceleration, braking, and handling, down to cost management and reliability. Therefore, the fastest car does not guarantee victory. The broad scope of the competition is designed to function as an optimal learning environment for teamwork, a skill deemed essential for engineers. [1]

History

The birth of Formula Student is dated to year 1980, when Assistant Professor Ron Matthews established a student branch of "Society of Automotive Engineers" (SAE) at the University of Texas. Shortly after, the idea of student competition was envisioned. The objective of which was to construct and race a formula-style four-cylinder combustion car. The first event was held only one year later. This concept subsequently gained global traction, which is still growing to this day. [2]

1.1 TU BRNO RACING

TU Brno Racing is a Formula Student team operating at the Brno University of Technology. The first season that the team participated in was fourteen years ago at the 2010/2011 academic year, with their Dragon 1. Significant changes have occurred over time. In addition to the fact that the team established itself in the world rankings in respectable positions, the most significant milestone was undoubtably the transition from combustion engine to electric powertrain. This year, the team is undertaking a significant shift in its design philosophy once again, increasing the number of electric motors from two to four. [3]

Dragon E5

The 2024/2025 season car is, as mentioned above, a big step forward for the team. The major conceptual change brought new challenges, but also new opportunities for improvement. The implementation of an individual motor for each wheel enables the incorporation of torque vectoring, thereby enabling faster cornering and improvements in acceleration. Conversely, the problems faced in the first year of this design are numerous, beginning with the necessity for increased battery capacity, new suspension geometry, and complete redesigning of the wheels due to the accommodation of the motors inside them. The issue of brake overheating becomes critical in this context. The new design of the wheel assembly greatly reduces the already weak airflow, barely keeping the brake temperature within acceptable limits in older vehicles. In combination with the higher weight of the car this problem would have undoubtedly resulted in the overheating of the brake system resulting in brake fade, thermoelastic instability or boiling of the hydraulic fluid. Furthermore, the previous design of the air ducts incorporated an attachment to the wheel hub that is no longer feasible. Therefore, a new cooling system for the brakes was required.

2 TYPES OF BRAKES AND THEIR COOLING

Brakes are the main means of stopping any moving vehicle, therefore their flawless operation is expected. Two primary types of friction brakes exist: drum brakes and disc brakes. Both of these systems utilize different mechanical configurations to convert the kinetic and potential energy into heat. This heat is subsequently dissipated into the surroundings. This dissipation includes not only the convection into the air, but also the conduction into the other components of the brake system, including the brake caliper and hydraulic fluid, among others. This may result in overheating and subsequent failure due to the brake fade, thermoelastic instability, or boiling of the hydraulic fluid. [4]

2.1 DRUM BRAKES

Although drum brakes remain in use, this is mainly due to cost-cutting measures, simplicity and the larger contact area of the braking elements compared to their disk counterparts. Due to these characteristics, they are only suitable for a niche group of applications, including the rear axles of budget cars and large trucks. [5]

2.1.1 CONSTRUCTION

The enclosed system of a drum brake design consists of a drum that is firmly connected to the rotating wheel. Inside the drum are two brake shoes that press against the inner surface of the drum to stop the wheel from spinning, as illustrated in Figure 2.1. The brake shoes, typically mounted on a backing plate within the drum, are actuated by a hydraulic or mechanical mechanism that exerts an outward force on the brake shoes, thereby creating frictional contact with the drum. The friction material on the contact surface of the shoes is specially treated to reduce wear and enhance the friction coefficient, ensuring effective braking while minimizing degradation of the drum. [6]

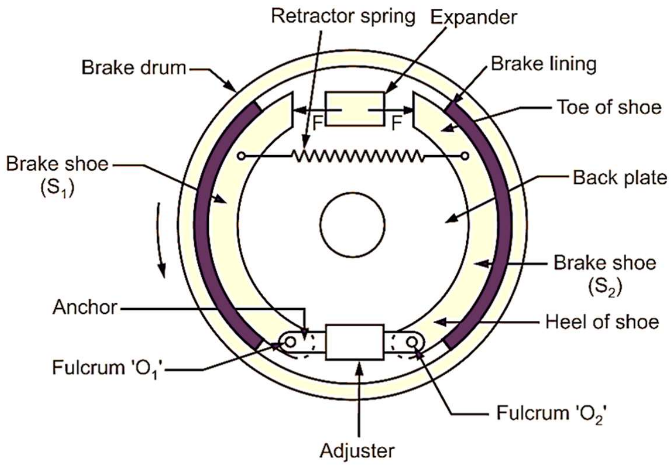

Figure 2.1: Internal Expanding Shoe Brakes (Drum Brake) [6]

2.1.2 COOLING

The enclosure of the entire brake system ensures the safe operation of this delicate assembly, shielding it from dirt and debris that would otherwise destroy it. However, this approach effectively prevents the cooling of the mechanism. In history, race cars were fitted with the largest available drums for the purpose of greater braking effect and better spread of the heat generated in the process. However, as time progressed, the drum brake design proved to be unable to satisfy the growing demands, ultimately leading to its widespread replacement by disc brakes, which possess significantly superior cooling capabilities. [5]

2.2 DISC BRAKES

Presently, there is an almost exclusive utilization of disc brake architecture. This trend may be credited to the relatively low weight, the higher power, the mentioned cooling potential, and enhanced, more predictable behavior, particularly in wet conditions. Thanks to rigidity of the exposed parts, the need for protective encloser is eliminated. The only components of the system that are exposed are the disc and the caliper body, which contain and protect the entire mechanism. [5]

2.2.1 CONSTRUCTION

The discs themselves rotate with the wheel and are exposed to the surrounding airflow, thereby cooling themselves throughout the revolution. Disc brakes are characterized by their adaptability to specific applications. The discs designed for automobiles are of a larger and more massive construction, whereas those intended for motorcycles are much thinner and lighter. The smallest discs are found on personal bikes and strollers. Brake discs are being fitted with drilled holes, grooves, and an inner cooling to further enhance the heat exchange. This subject will be addressed in subsequent paragraphs.

The braking mechanism in disc brakes is initiated by the application of pressure to the rear side of pistons via hydraulic fluid, which subsequently pushes the brake pads against the rotating disc between them. As illustrated in Figure 2.2, the hydraulic fluid is pressured into the sealed cavity situated behind the pistons, thereby exerting an outward force on them. After the pressure is relieved, pistons are retracted back inside the caliper body by the distorted seal ring. Pistons are composed of high-end materials that exhibit resistance to repeated heating and pressures. The pistons are hollow on the side, which is facing the brake disc, a design element that reduces weight while enhancing the heat isolation of the hydraulic fluid, thereby preventing its boiling. [4]

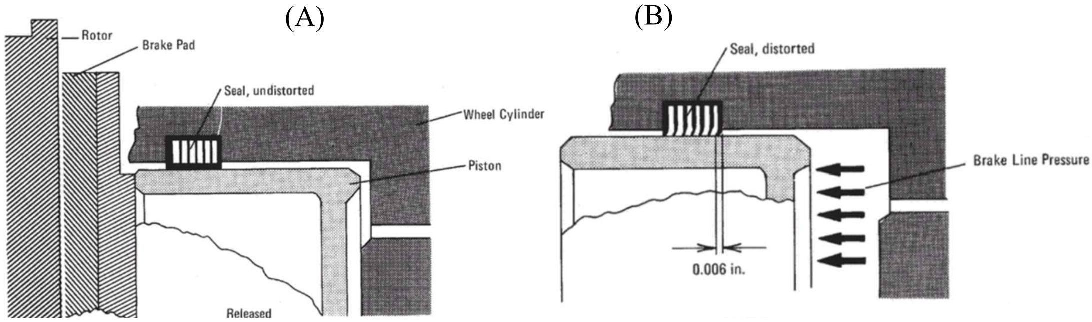

Figure 2.2: Disc brake adjustment. (A) Released brakes. (B) Applied brakes. [4]

2.2.2 MATERIALS

The composition of the discs varies, depending on the requirements. Cast iron discs are utilized in applications where cost is a significant factor, often at the expense of increased weight. Steel exhibits superior resistance to higher temperatures at smaller thicknesses. However, it is susceptible to bending. Laminated steel, while boasting enhanced durability, is comparatively rare, with production limited to small series for application in racing. Aluminum alloys are mainly employed in applications that do not demand the same level of performance as other materials, due to their lower melting point and inferior abrasion resistance properties. Carbon-ceramic brake discs represent the high-performance option. These discs are composed of cut carbon fibers in a ceramic matrix, a composition that contributes to their high cost, making them an optimal choice for the toughest of applications. The carbon-carbon composition differs from the previous one only in that it contains a graphite matrix, which contributes to even lighter discs for the most niche fields. The high performance exhibited by carbon-ceramic and carbon-carbon brake discs is rendered less practical in many applications due to their challenging operation, which requires maintaining temperatures within a precise optimal range, a factor that, combined with their significantly higher costs, limits their widespread use despite their advanced capabilities. [5]

2.2.3 COOLING OF DISC BRAKES

Cooling of the brake assembly plays a crucial role in high demand scenarios like Formula Student races. Without sufficient cooling of the brake discs, there is an increased probability of permanent deformation. Consequently, this results in the formation of "hot spots," which can lead to the development of thermal cracking. Excessive temperatures of the brake calipers can also lead to problems, namely the overheating of the hydraulic fluid to its boiling temperature can reduce or even terminate the braking effect. Another problem, occurring primarily with solid brake discs without any slots or holes, is brake fading. This term refers to the process of gas accumulation during the braking process. These gases interfere with the contact between the pad and the disc, thereby compromising performance. Therefore, it is critical to maintain the system at its operational temperature. [4]

Slotted brake discs

The most straightforward method for enhancing the brake disc capability of heat dissipation is to incorporate grooves or holes onto the surface. The surface grooves, that can be seen in Figure 2.3, serve a dual purpose: they enhance heat dissipation and, secondly, they cut into the brake pad material, thereby renewing the top layer of the material with each revolution. This process eliminates the old, used material and also effectively expels the gases created by braking delaying the brake fading, ensuring the best brake conditions while the brake pedal is engaged. [7]

Drilled brake discs

The second option for enhancing the performance of brake discs involves the implementation of drilled perforations in the disc. The primary benefit of this approach is its superior heat dissipation capabilities, which surpass those of the previously mentioned slots. However, the lack of a "pad regenerating feature" and the lower effectiveness in preventing the accumulation of dirt, water, and gases are notable drawbacks. Furthermore, the presence of holes in these discs can contribute to thermal cracking due to the accumulation of stress on their edges. [5; 7]

Figure 2.3: Drilled brake disc on the left, variations of the slotted brake discs in the middle and on the right [7]

Ventilated disc brakes

Another solution to improve the heat dissipation and performance over extended periods of time is the integration of cooling channels along the radial direction of the disc, an approach that can be employed in conjunction with the previously outlined options. This configuration allows the heat to be dissipated into the ambient air more effectively due to the increased surface area. Despite the fact that the majority of the heat is mostly accumulated on the sides of the disc, as can be seen in the cutout in Figure 2.4 (where Ti is designated as the ambient temperature), this technology is highly effective and is widely used nowadays.

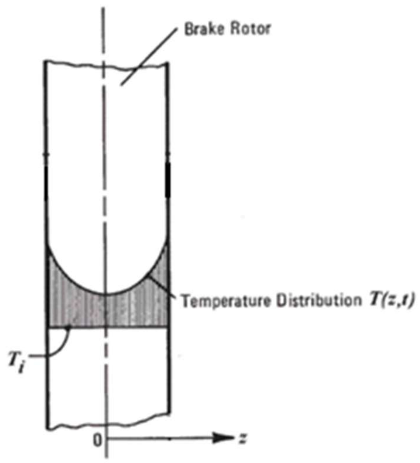

Figure 2.4: Simplified cutout representing temperature distribution in solid rotor [4]

As previously mentioned, thanks to the increased surface area, the convective heat transfer coefficients approximately double in size compared to their solid disc counterparts. In practice, the fresh and undisturbed air as possible is directed into the wheel close to its hub, by various channels and ducks, as can be seen on Figure 2.5. Subsequently, the air is drawn into the disc by the vanes. In high-performance applications, such as Formula 1, these vanes are replaced by more than a thousand small, precise holes. This results in enhanced cooling efficiency and an increase in price. [4; 8]

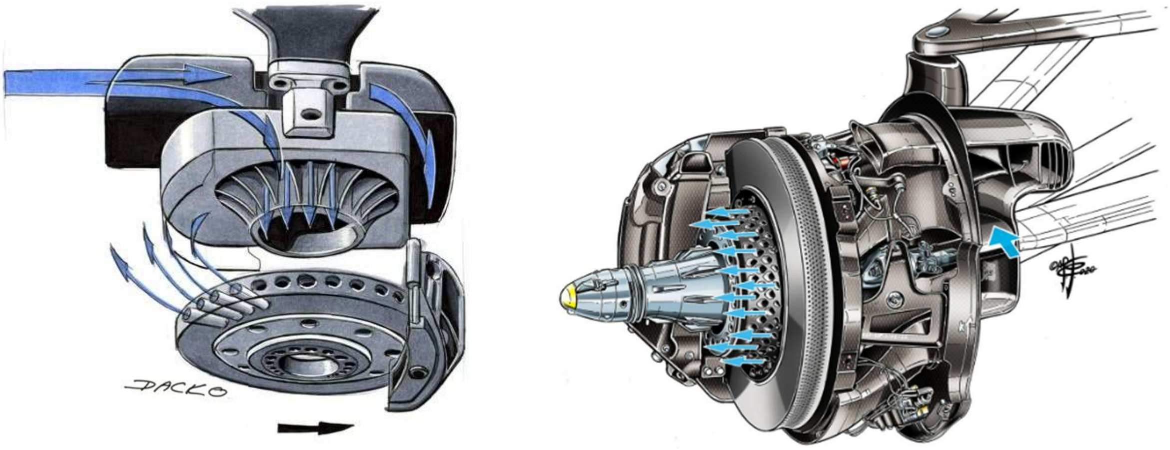

Figure 2.5: Ducts directing air to cool the brakes [14; 15]

2.3 MOTORCYCLE BRAKES

Motorcycles are much lighter than automobiles; therefore, they exhibit reduced levels of kinetic energy. Consequently, the utilization of heavy brake discs with inner cooling mechanisms would not be effective. Much lighter and narrower discs are being used. In contrast to the automobile brake discs, where slots are exclusively surface modification, there is a wide-spread utilization of drilled through slots in motorcycle brakes. These modifications result in a further reduction in weight and moment of inertia. A diverse array of designs combining holes and slots is being used, with the overall appearance of the motorcycles being taken into consideration. [5]

2.3.1 FIXED-CALIPER DISC BRAKE

This design is predominantly utilized in combination with the floating brake disc, which is composed of an outer friction part that is connected to the center piece by pins. The outer part can be axially defined by plate springs or can be unrestricted, thereby permitting axial movement between the contact pads of the caliper. This configuration enables the material to expand under heat, thereby significantly reducing strain on the component. Fixed calipers are most commonly equipped with one or two pistons on each side, as is the case in Figure 2.6. These pistons transfer the pressure from the hydraulic fluid to the pads, pushing them against the disc. The body of the caliper is fixed to the front fork of the motorcycle in this configuration. [5; 4]

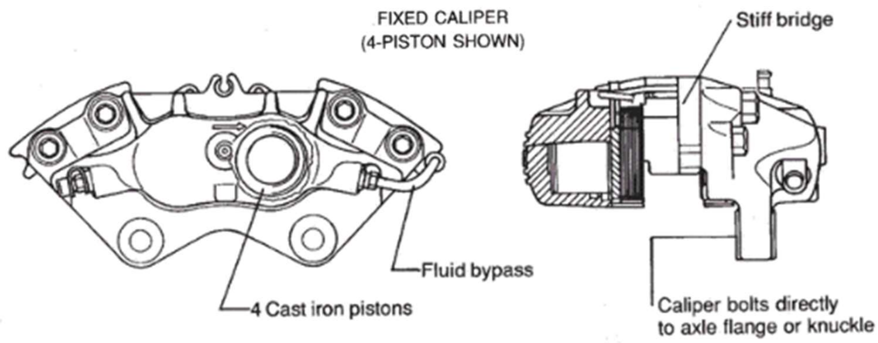

Figure 2.6: Fixed caliper design with four pistons [4]

2.3.2 FLOATING-CALIPER DISC BRAKE

In contradistinction to the fixed design, floating calipers are exclusively paired with discs, which are axially fixed. Therefore, it is necessary for the caliper to be able to move on the pins, which are fixed to the motorcycle. These calipers are distinguished by their single- or double-piston design, with the pistons positioned on a single side of the disc, as illustrated in Figure 2.7. Applying pressure to the system causes the pistons to move outwards, thereby forcing the pad into contact with the brake disc. Subsequently, the pressure displaces the whole body of the caliper moving the second pad into contact. [5; 4]

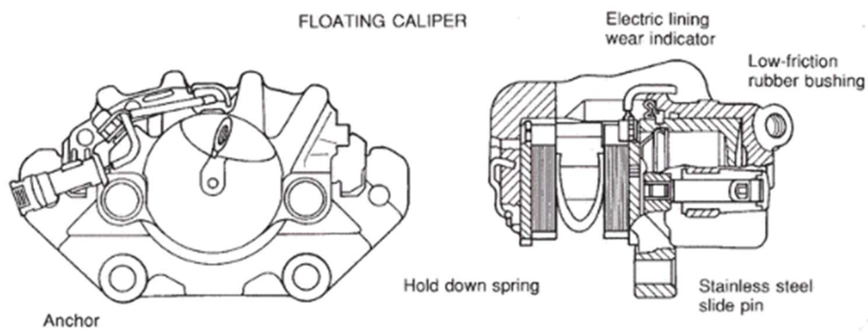

Figure 2.7: Floating caliper design with one piston [4]

2.4 BRAKE SYSTEM ON THE ED5 FORMULA STUDENT MONOPOST

For a considerable number of teams participating in Formula Student, the most straightforward option for their brake system design is to purchase an off-shelf motorcycle brake disc with all the accessories, such as brake calipers. For teams with more experience and better funding, the more advantageous option is to manufacture their own motorcycle-like brake system. The decision to employ motorcycle brake systems is often made, thanks to their compact size, which allows the teams to neatly pack them within the small wheels (10–13 inches) used in FS cars ensuring a low center of gravity. These brakes are also sufficient thanks to lightweight nature of the cars (approximately 150–300 kilograms) that compete in FS events.

In the upcoming season, the TU Brno racing team will be manufacturing their entire brake system in-house for the first time. The team has been manufacturing its own brake discs for an extended period. However, with the 24/25 season, there is expected to be a transition in the caliper utilized, from purchased Brembo calipers to in-house designed and manufactured ones. Following numerous design iterations, the brake discs from previous years were equipped with expanding drilled through grooves, arranged in a pattern resembling a portion of a spiral, oriented in the direction of the forward rotation of the wheel, as can be seen in Figure 2.8, to reduce the bending moment arm from the frictional force. [5]

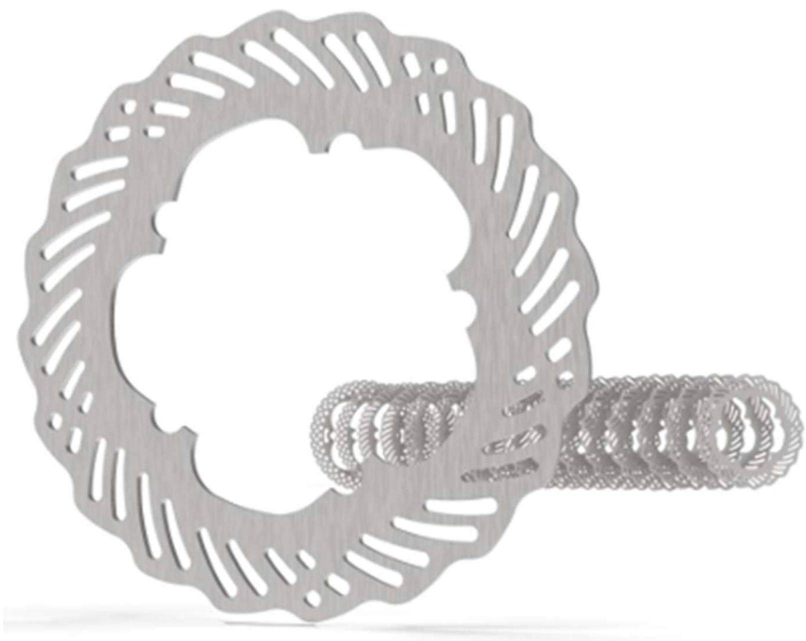

Figure 2.8: Final design of the ED2 brake disc and its iterations [5]

The grooves are also designed to enhance airflow through the disc, thereby improving convective cooling and reducing peak temperatures during braking. Due to the installation of the four in-wheel electric motors on the new ED5 the brake discs needed to be redesigned in the area where they are connected to the wheel hub. Furthermore, given the increased weight of the vehicle in comparison with previous models, the new brake discs will be drilled with smaller holes, adopting a cautious approach with the potential for future optimization, due to concerns about the higher demands on the brake system.

The focus of this thesis is the development of a cooling mechanism for the new brake system. The objective was achieved through the design of brake ducts that guide the ambient air into the wheel assembly. The designed ducts are made out of fiberglass, a material that ensures electrical nonconductivity and aligns with the competition's rules. The ducts are attached to the hub by a bracket made of aluminum, which is seamlessly incorporated into a preexisting part of the motor assembly, thus reducing the added weight of the attachment. While the integration of brake ducts has been demonstrated to enhance the cooling effect (a topic that will be discussed in the following section), the design and implementation presented challenges. The placement and shape of the ducts must not have interfered with other components, such as the suspension and aerodynamic elements. These limitations ultimately determined a significant portion of their appearance, which can be seen in Figure 2.9.

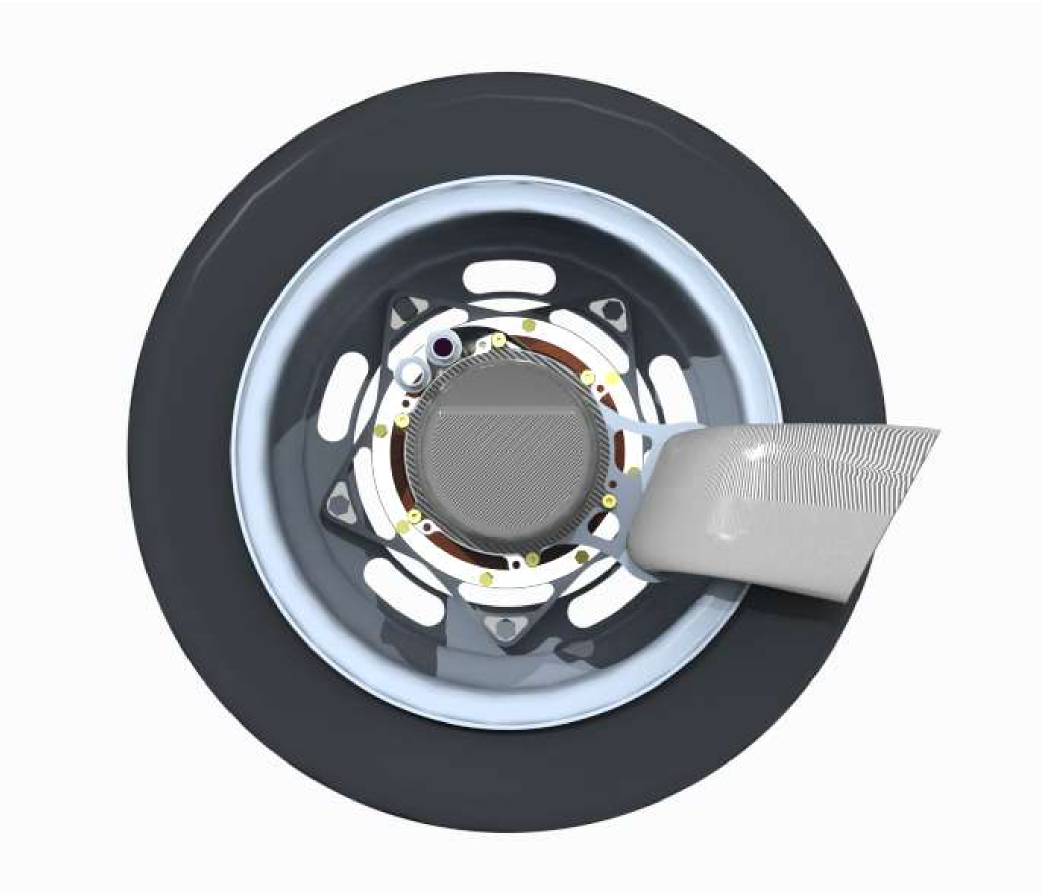

Figure 2.9: Clarified view on the brake duckt and its attachement to the wheel hub by the bracket

3 CFD SIMULATION

Computational fluid dynamics (CFD) simulation is a numerical method used to analyze complex problems related to laminar and turbulent flows, heat distribution, heat dissipation, and other multiphysics phenomena. In the context of this thesis, this approach was employed to analyze the thermal behavior and of the brake disc system, particularly under high-load conditions like those experienced during Formula Student races. For the purposes of this simulation, the Ansys Fluent software was utilized.

3.1 GEOMETRY

In the pursuit of enhanced speed and precision of simulations, geometry is often simplified for. In this case it was unnecessary to include neither chassis, suspension rods, nor the rear half of the car. The components' dimensions and the distance over which they are situated have a very negligible effect on the airflow and heat transfer inside the wheel. Furthermore, the electric motor and its transmission were simplified into cylinders due to their complex shape. Unfortunately, the brake disc itself must have also undergone simplification for the purpose of this thesis in order to maintain its rotation in the simulation. The disc was modeled as a solid object, excluding the presence of holes or slots. The final geometry used in the CFD is illustrated in Figure 3.1.

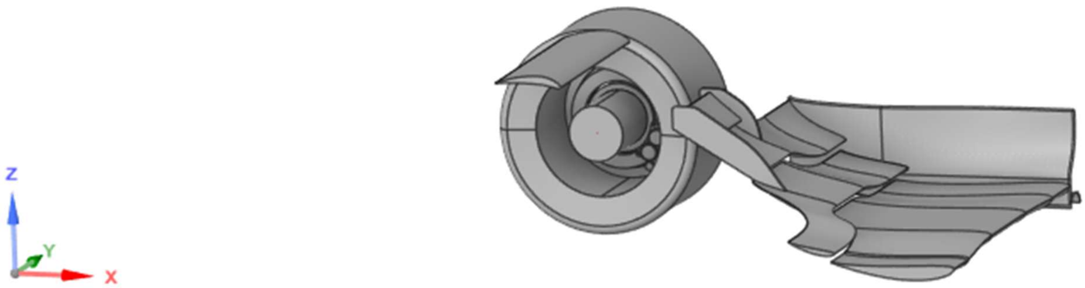

Figure 3.1: Simplified geometry used in the simulations

The simulation will be conducted four times in total, three times with different outlet positions and lengths marked A, B, C, as can be seen in Figure 3.2, and one time without the duct. The bracket that holds the brake duct in place will not be incorporated in the geometry model. However, as illustrated in Figure 3.2, the location for this bracket connection is preserved throughout the designs.

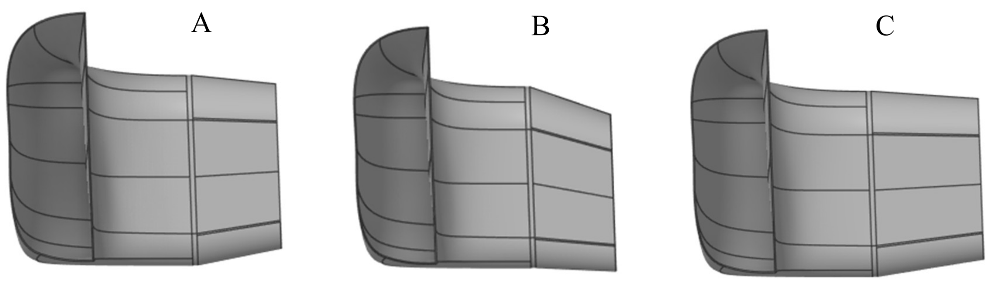

Figure 3.2: Different designs of brake duct outlet used in simulations

The brake caliper, a critical component of the simulation, was modeled with high precision in Ansys SpaceClaim to ensure accurate thermal behavior. Only minor adjustments were made to improve mesh quality in certain geometrically complex regions. Figure 3.3 shows a cross-sectional view of the caliper and disc assembly, with numbered labels and color coding to identify each component's material. Specifically, steel is designated by a gray color, brake pad material by a red color, air by a blue color, titanium by an orange color, hydraulic fluid by a green color, and aluminum by a purple color. The steel brake disc (1) is positioned at the center of the assembly and comes into contact with the brake pads (2) during braking. The pads are mounted on steel backing plates (3), which are actuated by titanium pistons (4) to generate the brake force. Behind each piston is a narrow gap, filled with hydraulic fluid (5), which functions as a medium for pressure transmission from the brake system. The most exterior element is the aluminum caliper body (6). The blue regions represent air pockets within the pistons (7). In practice, these pockets serve the purpose of reducing weight and providing insulation for the caliper and hydraulic fluid from the high temperatures.

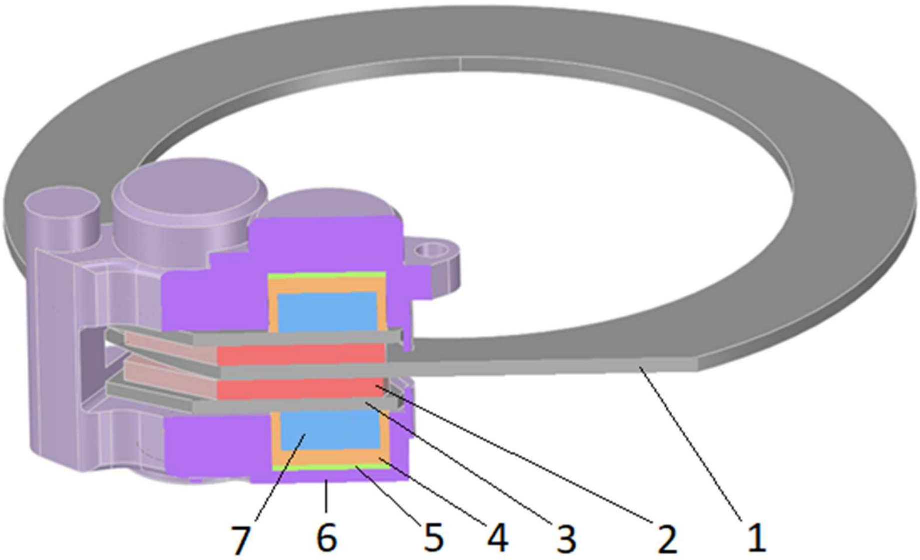

Figure 3.3: Section of the brake assembly used in simulation

3.1.1 DOMAIN

The airflow around the geometry was modeled using a basic rectangular fluid domain, displayed in Figure 3.4. In order to optimize computational efficiency and reduce mesh size and cell count, the simulation leveraged the symmetry of the vehicle design, focusing solely on the left side of the vehicle. This enabled detailed analysis of the heat exchange within the brake caliper and its vicinity. A key element of the simulation was the implementation of shared topology, a mechanism that guaranteed a conforming volume mesh across the fluid and solid domains. This is a critical element in the simulation of heat exchange between brake components and the surrounding air flow. This was achieved in the preprocess in Ansys SpaceClaim, where the domain was defined with shared topology by aligning the interfaces between the disc, caliper, and fluid region.

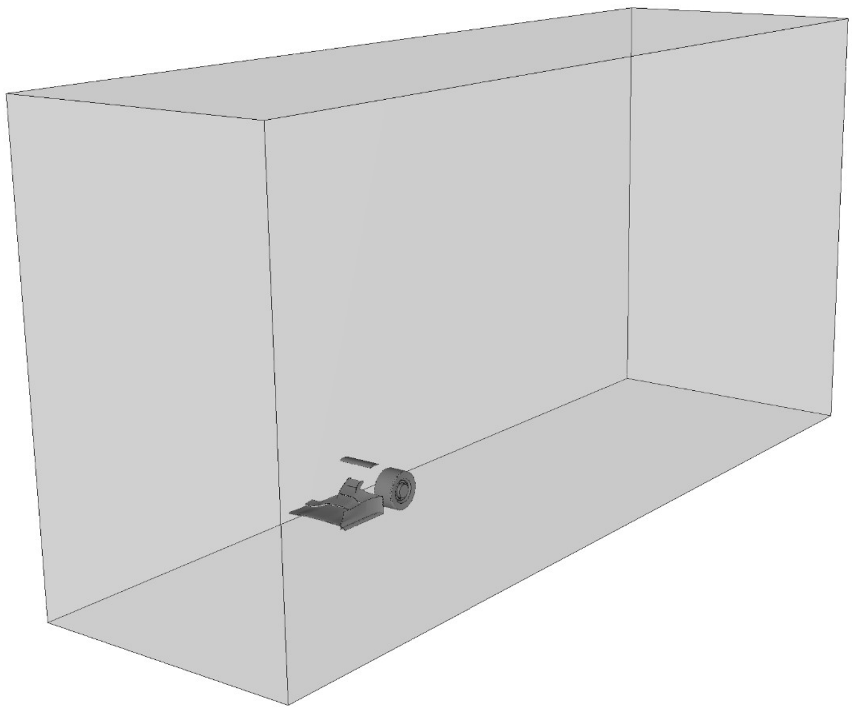

Figure 3.4: Computational domain

3.2 COMPUTATIONAL MESH

The computational mesh is a fundamental component in achieving accurate and reliable solutions within computational fluid dynamics (CFD) simulations. It serves as the discrete framework upon which the governing equations of fluid flow, heat transfer, and thermal dissipation are numerically solved. The mesh is composed of a finite number of small cells, or control volumes, where the Navier-Stokes equations, energy equation, and turbulence model equations are discretized and solved. [9]

3.2.1 SURFACE MESH

The meshing process begins with the creation of a high-quality surface mesh, a critical step to ensure accurate thermal analysis. A well-defined surface mesh is essential for capturing the complex geometry of the brake system, including the surfaces of the wheel and the front wing of the car affecting the airflow and the interfaces between components, thereby directly influencing heat transfer. The whole operation of mesh generation was done in Ansys Fluent, making use of the watertight geometry workflow method. To address challenging areas prone to mesh distortion, two types of localized sizing were implemented: curvature sizing and proximity sizing. These techniques further improved the quality of the mesh quality and ensured reliable simulation results.

Curvature sizing

Curvature sizing is a method that uses maximal and minimal cell size, in addition to normal angle, to refine the surface mesh on curved surfaces. Figure 3.5 shows the progressive refinement of the mesh in proportion to the curvature of the surface. [9]

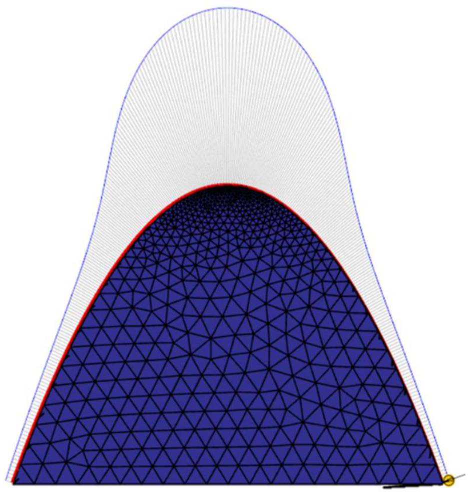

Figure 3.5: Mesh fitted with curvature sizing[9]

Proximity sizing

Proximity sizing is a method that ensures that the minimal number of cells is generated between two edges. This approach helps prevent distortions of the geometry in these areas and enables the overall minimal cell size to be bigger, thereby reducing the total cell count. It is considered a best practice that at least three cells are generated over the width of any given wall. [9]

3.2.2 VOLUME MESH

The next step in the preparation of the simulation is the generation of a volume mesh on the preexisting surface mesh. For this simulation, the "poly-hexcore" cell setup was utilized. This approach generates polyhedral cells in proximity to the geometry, thereby enhancing precision, and cube cells on the waist spaces for streamlined calculation. In the immediate vicinity of the surfaces, as well as on the surfaces themselves, the fluid generates a boundary layer. This boundary layer must be incorporated into the simulation through the implementation of prismatic layers.

Prismatic layers

The formation of the boundary layer occurs due to the zero air velocity on the geometry surface, caused by the viscosity of the air. The accurate modeling of the change in speed requires the implementation of sufficiently small cells. This objective is realized through the implementation of prismatic layers.

The setup of prismatic layers is initiated with the establishment of the generation method within the watertight geometry workflow. In this simulation, the uniform offset method was selected for this simulation, given its ability to directly manipulate the initial layer height. This, in combination with an adjustable number of layers, allows for an uncomplicated pathway for enhancing the mesh quality. This method ensures that the thickness of each layer remains constant throughout the geometry. However, the drawback of this approach is the inability to regulate the transition between the prismatic layers and volume mesh, and that it is susceptible to deformation when applied to sharper angles. Due to this the setup may require additional iterations to properly model the boundary layer, as is illustrated in Figure 3.6.[10]

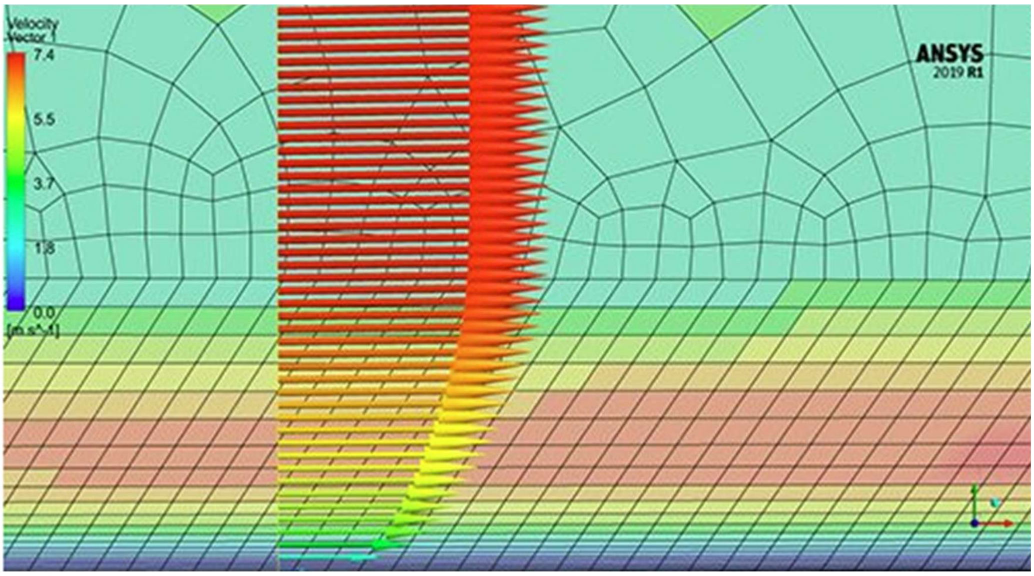

Figure 3.6: Properly modeled prismatic layers [16]

Quality of volume mesh

It is important to thoroughly examine the generated volume mesh prior to simulation in order to ensure the seamless execution of the simulation and the reliability of the obtained solutions. A proper evaluation should consist of visual assessment of geometry, as by the time the volume mesh is generated, the geometry takes on the shape of the cells surrounding it. Therefore, an examination of the curves and edges is necessary to ensure that there has been no significant deformation of these surfaces. The second step in the mesh validation process should be orthogonal or skewness quality control. The implementation of a task designed to enhance the volume mesh is a viable option within the watertight workflow. Orthogonal quality primarily measures a metric that detects poor cell shape at a local edge, such as twisting and/or concavity, in addition to other properties of the mesh. Skewness, on the other hand quantifies how far from ideal (that is, equilateral or equiangular) a face or cell is. The minimal orthogonal quality should be approximately 0,15 for the orthogonal method or 0,85 for the skewness method. [11]

3.3 MODEL DEFINITION

The model definition establishes the physical and mathematical foundation for the CFD simulation necessary to represent the complex interplay of fluid dynamics and heat transfer. The following paragraphs detail the approaches and methods utilized in this simulation to model the behavior of the brake system under load.

3.3.1 VISCOSITY MODELS

There are several methodologies for modeling turbulent flow in CFD simulations. The most accurate, but also the most demanding method is DNS (Direct Numerical Solution), in which the Navier-Stokes equations are solved directly. Less demanding option and much widely used one is LES (Large Eddy Simulation). This method solves only large vortices directly, while the small ones are modeled. The most common option, despite its lower accuracy, is RANS (Reynold-Averaged Navier-Stokes). This method uses time-averaged metrics, that enable the use of static simulation. A new term, in the form of the Reynolds stresses, must be added to Navier-Stokes equations, when utilizing the RANS method. The Reynolds stresses are solved separately in additional equations with the aid of turbulence models to properly model the flow. [9]

Model k-omega SST

The Shear Stress Transport (SST) k-omega turbulence model, employed in the simulation of the brake system, combines the strengths of the k-omega and k-epsilon turbulence models to achieve improved accuracy in predicting turbulent flow. The k-epsilon model, a widely used two-equation model, performs well in free-stream regions but lacks the accuracy to predict flow separation and adverse pressure gradients. In contrast, the k-omega model excels in resolving near-wall flows and flow separation, making it suitable for capturing the boundary layer effects critical to heat transfer in the brake system. The SST k-omega model leverages a blending function to integrate the k-omega formulation near walls with the k-epsilon formulation in the free stream. This hybrid approach ensures accurate modeling of turbulent airflow and convective heat transfer around the brake components. [9]

3.3.2 ENERGY EQUATION

Due to the necessity of incorporating heat transfer in the simulation, it is necessary to add an additional equation. This means that Fluent additionally solves the RANS-averaged energy equation, which is derived from the Navier-Stokes energy conservation equation. The turbulent heat flux is modeled using the turbulent viscosity and the turbulent Prandtl number, which are influenced by the SST k-omega model. [11]

3.3.3 MATERIAL CHARACTERISTICS

The thermal simulation requires the definition of material characteristics for both the solid and the fluid domains. These properties include density (ρ), specific heat capacity (c_p), thermal conductivity (λ) and the viscosity (μ) in the case of the fluid domain. The values for these properties can be found in Table 3.1 and Table 3.2. The aluminum, steel, and titanium values were obtained directly from the Ansys fluent database, ensuring reliable and standardized values. Due to the absence of required data in the database, the properties of the brake pad material were manually defined. However, accurate values are difficult to obtain, therefore, an estimate had to be made. In this instance, the flow of air within the pistons and of hydraulic fluid was not taken into consideration. This simplification was deemed acceptable due to the fact that conduction is the primary heat transfer path, as the limited volumes restrict significant convective heat transfer. The properties of air for the external fluid domain were obtained from the Fluent database. Additionally, all material properties needed to be set for each domain as a cell zone condition.

Table 3.1: Air characteristics from Ansys fluent database

	ρ - density $[kg/m^3]$	c_p - specific heat $[J/(kg \cdot K)]$	λ - thermal conductivity $[W/(m \cdot K)]$	*μ - viscosity* $[kg/\mathrm{m} \cdot \mathrm{s}]$
Air	1.225	1006.43	0.0242	1.7894e-05

Table 3.2: Solid material characteristics used in simulation

	ρ - density $[kg/m^3]$	c_p - specific heat $[J/(kg \cdot K)]$	λ - thermal conductivity $[W/(m \cdot K)]$
Air	1,225	1006.43	0.0242
Aluminum	2719	871	202,4
Hydraulic fluid	900	1900	0,15
S33 Sintered metal-ceramic	2500	800	1,5
Steel	8030	502,48	16,27
Titanium	4850	544,25	7,44

3.4 BOUNDARY CONDITIONS

The boundary conditions specify the flow and thermal variables on the boundaries of each domain. They are therefore a critical component of simulations, and it is important that they are specified appropriately. [11]

Frame motion

Frame motion, otherwise referred to as moving reference frame (MRF), is a steady-state approximation used to model rotation without physically moving the mesh. This is achieved through the introduction of a rotating reference frame, which modifies the equations to incorporate additional acceleration terms on the domain interfaces. This is an effective method for studying flow around geometry, that rotates in real world applications, without the need for demanding transient simulations. A notable disadvantage of this approach is that the geometry itself does not rotate. Therefore, this approach is not perfectly suited for simulating scenarios studying the impact that the influenced flow from the rotating components has on other parts. In this simulation, the MRF approach was used to model the effects of the wheel rim's rotation on the surrounding and passing airflow. This region is illustrated in Figure 3.7 with the color blue. [11; 12]

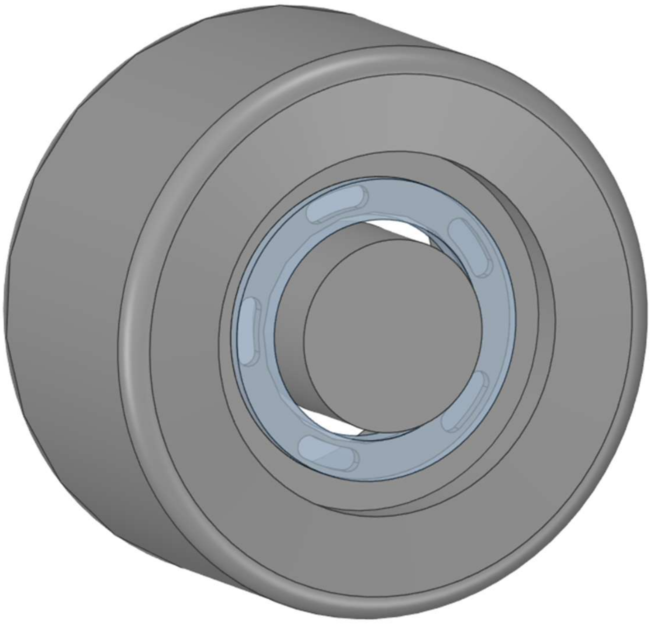

Figure 3.7: MRF region around the wheel rim

Wall motion

The final option employed for modeling rotation in this simulation was wall motion. This boundary condition specifies the tangential velocity of the interface wall. As the name suggests, the influence of this boundary condition is confined to the interface between the domains, making it the least demanding option for simulating rotation. However, it is imperative to approach this condition with a degree of caution. As previously mentioned, this approach applies only tangential motion to the interfaces. This may result in inaccuracies in the modeling of motion, which occur in the normal direction, for example, rotation of the spokes inside a wheel. In this simulation, the wall motion boundary condition was used to emulate the movement of the ground and the tire. [11]

Solid motion

The solid motion approach was introduced to the simulation due to the fact that the MRF approach is not viable for the simulation of the brake disc rotation. This is due to the fact that the MRF mechanism does not permit the movement of the brake disc relative to adjacent domains (the adjacent domains to the brake disc in this simulation are the domains of brake pads). The solid motion approach, as in the case of the wall motion, requires the velocity to be divergence-free, which means that the specified motion will always result in velocity tangential to the boundaries. Therefore, the geometry of the brake disc must have been simplified to an annular disc, eliminating any faces perpendicular to the motion. The effects of motion inside the solid zone are applied only to the energy equation, yielding realistic temperature dissipation during rotation without the use of complex transient simulation. [11]

Thermal conditions

In CFD simulations involving heat transfer, as previously stated, the energy equation must be enabled in Ansys Fluent to solve for temperature distribution across all domains. Additionally, thermal boundary conditions must be carefully defined to accurately model heat transfer between domains, such as conduction within the brake disc and convection to the surrounding air. [11]

When a mesh contains multiple domains in contact Ansys Fluent automatically generates a so-called two-sided wall at the interface. A two-sided wall consists of two overlapping wall boundaries, one belonging to each domain, which are treated as separate entities by default. This allows different thermal and motion conditions to be applied on each side of the interface. This approach was not optimal for the analysis of continuous heat dissipation through the brake caliper. To address this, Fluent provides the option to use a coupled thermal boundary condition, which ensures that the temperature and heat flux are continuous across the interface. In this simulation, the coupled thermal boundary condition was applied to all interfaces, including the brake pad-disc interface, to ensure accurate modeling of heat distribution into the brake disc and the caliper assembly. This approach is particularly important for capturing the convective cooling of the whole assembly by the airflow, which influences thermal failure modes, such as brake fade and thermoelastic instability. [11]

To simulate the heat generated during braking, a heat source was applied at the interfaces between the brake pads and the disc. This required defining the wall thickness, wall material, and heat generation rate. The wall thickness was set to 0,1 millimeters to approximate the thin contact layer between the brake pads and the disc, creating a virtual volume at the interface for applying the heat source. The wall material was specified as the brake pad material, with properties defined earlier in Table 3.2. Finally, the heat generation rate was calculated based on a heat flux derived from telemetry data from testing of an older model of a TU Brno Racing monopost. The telemetry analysis, which recorded brake pedal force and vehicle deceleration during autocross events, determined that the power output to the front brake disc was 1986 W. To convert this power into a heat generation rate, the contact area between the brake pad and the disc was measured in Ansys SpaceClaim, the value equaled to 1080 mm^2. The heat generation rate ($\dot{q}(t)$) at the interface was then calculated using the following formula:

$$\dot{q}(t) = \frac{P}{2 \cdot A \cdot b} \tag{3.1}$$

, where P is the power output to the front brake disc, A is the surface area, and b is the wall thickness. For a wall thickness of 0,1 millimeters, the heat generation rate is calculated to be $\dot{q}(t)$ = 9,2 GW/m^3. This heat generation rate was applied to the pad-disc interfaces to simulate the heat produced during braking. The heat flow value is specific to the telemetry data from a previous vehicle and may not perfectly match the ED5's braking conditions due to differences in vehicle dynamics, wheel hub design, and weight. However, it provides a realistic estimate for evaluating the thermal performance of the brake system and the designs of the brake ducts in the simulation. [11]

3.5 SOLUTION STRATEGY

The solution strategy employed for the CFD simulation was selected with the objective of ensuring robust convergence and accurate results.

3.5.1 SPATIAL DISCRETIZATION

In the Ansys Fluent, discrete values are stored at the cell centers. However, for the purposes of convection, face values are required. Therefore, upwind schemes are employed. This process derives the face value from the quantities in the cells upstream, relatively to the flow, hence their name. First-order and second-order upwind schemes are adequate for a wide range of applications, including the case at hand. For more specialized applications, the QUICK and third-order MUSCL schemes are also possible approaches. However, as they were not deemed suitable for this particular simulation; therefore, they will not be further elaborated upon. [9; 11]

First-order upwind scheme

The first-order upwind scheme uses an assumption that the values at the center of the cell represent the cell average and that these values are consistent across the faces. Hence, the face value is equal to the discrete value at the center of the upwind cell. This method is feasible only if the flow is aligned with the mesh, a condition that is not always met in this case. [9; 11]

Second-order upwind scheme

For meshes such as triangular and tetrahedral, the utilization of a second-order upwind scheme will result in more accuracy. This is achieved through a Taylor series expansion of the cell-centered solution about the cell centroid. Due to this added complexity the calculation duration and instability are increased. However, this is counterbalanced by an improvement in precision. The employment of this particular method of spatial discretization was justified, due to the presence of polyhedral cells that do not always align with the flow and the importance of accurate simulation in these regions. [9; 11]

3.5.2 PSEUDO TIME METHOD

Steady-state solvers are inherently prone to divergence due to the nonlinear nature of the governing equations, necessitating stabilization techniques to ensure convergence. A fundamental approach in Ansys Fluent is the use of under-relaxation factors (URFs), which are applied to all primary variables. This approach serves to restrict the magnitude of updates in each iteration, thereby enhancing stability. However, relying solely on relaxation can lead to slow convergence, and determining optimal URF values for complex problems can be challenging. To address this, the pseudo-transient simulation method was employed, which enhances the stabilization process by working in conjunction with the existing relaxation factors. This approach introduces an artificial time derivative, referred to as pseudo time step size, designated as $\Delta\tau$, into the algebraic form of the steady-state equation. By incorporating the pseudo-time step, the method modifies the effective relaxation behavior, adjusting the update magnitude based on the flow field characteristics. The pseudo time step size $\Delta\tau$ is determined through the application of either the local or global time step method. [11; 13]

Local time step method

The local time step method involves the calculation of distinct local pseudo time steps for each cell, based on the provided pseudo time Courant number and the flow scales. This under-relaxation approach is recommended for better convergence at the expense of conversion rate. [11]

Global time step method

The global time step method, as its name implies, utilizes only a single pseudo time step size for the entire simulation domain. The pseudo time step size can be manually selected or can be calculated automatically, with further possibility of modification by specifying a time scale factor. It is possible to set different time scale factors for solid and fluid domains, resulting in different pseudo time step sizes for these domains. The implementation of pseudo time step is also an effective strategy for accelerating the simulation process. In general, the convergence is superior when the time step is reduced. However, for the intermediate solutions, where the heat is being slowly transferred to further parts of the caliper and the disc, larger time steps significantly reduce the calculation time, as shown in the Figure 3.8. [11; 13]

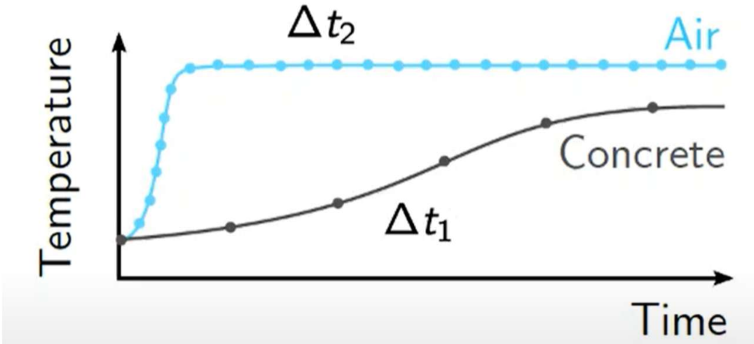

Figure 3.8: Different step sizes in solid and fluid domains [13]

3.6 INITIALIZATION

Prior to the engagement of any CFD calculation, it is important to provide an initial estimate of the solution flow field to the solver. This process is crucial in reducing the calculation time necessary to obtain valid results. There are three distinct approaches to initialization that can be employed to achieve this objective. [11]

Standard initialization

The first possible approach is standard initialization, in which the initial values, such as flow characteristics, are manually set by the user. These values are subsequently assigned to all cells. This approach is fast, and is typically applied to relatively simple geometries and due to this. However, this method has been deemed suboptimal for this particular simulation due to its inferior performance in comparison to hybrid initialization. [9; 11]

Full multigrid initialization

FMG initialization, the second initialization option, uses a multigrid approach, solving the governing equations on a series of progressively larger grids before interpolating the solution back to the fine mesh. This approach provides a more accurate initial estimate compared to a standard or hybrid initialization described below. However, the accuracy of the initial estimate is accompanied by a longer computation time. A notable disadvantage of this approach, which is also the reason behind the decision to not utilize this particular approach of initialization, is the necessity of a uniform initial solution. This solution can be obtained through standard or hybrid initialization, or by inserting specific data. [11]

Hybrid initialization

The final approach is hybrid initialization, which is a collection of methods and boundary interpolation schemes to establish a robust initial solution for the CFD simulation. This method provides an initial conformal velocity and pressure field over the domain and its geometries. Subsequent variables, including turbulence quantities and temperature fields, are then determined based on spatially averaged values derived from the boundary conditions and domain characteristics. This method is widely applicable and was used for this simulation. [9; 11]

4 RESULTS

It is important to note that all simulations were conducted with the assumption that the vehicle is traveling in a straight trajectory at a velocity of 60 kilometers per hour. This speed was selected because it aligns with the generally lower average speeds typically seen in Formula Student courses. Additionally, this speed ensures compatibility and comparability with other simulations executed by the team TU Brno Racing, aligning with the established practice of this team.

The credibility of the simulations was assessed based on two conditions. One of them is the visual inspection of the results, as illustrated in Figure 4.1. This figure clearly demonstrates a gradual decrease in brake disc temperature, approximately 5 Kelvin, during one revolution.

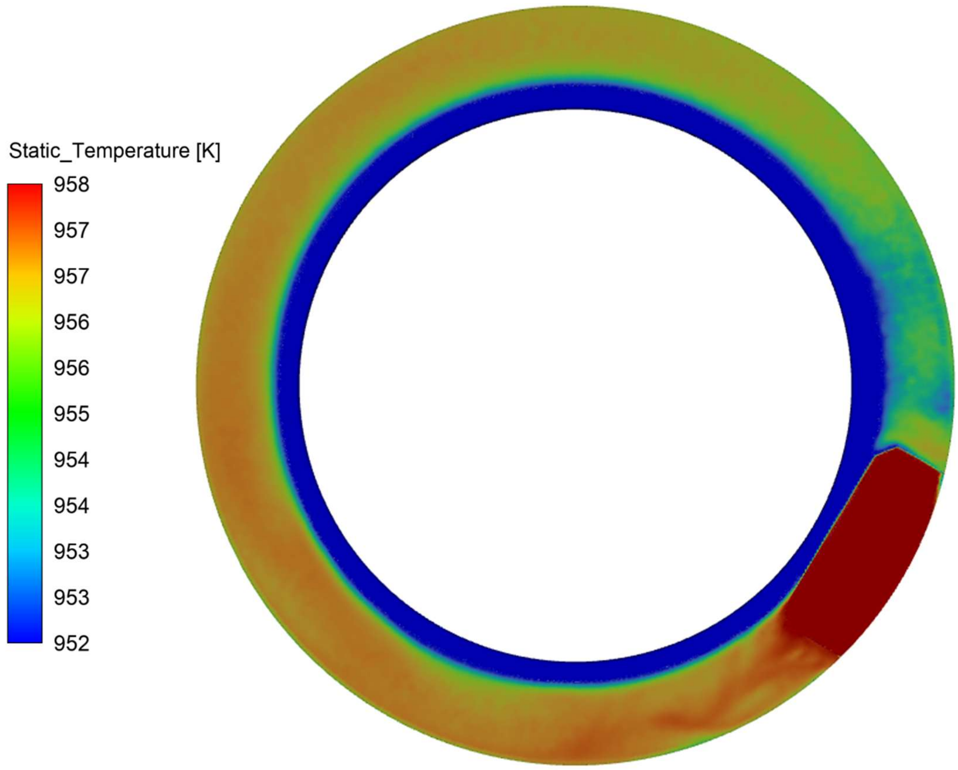

Figure 4.1: Heat disipation on the brake disc throughout the rovolution

The second condition consisted of a comparison of the anticipated surface temperature of the caliper, derived from prior measurement data, with the temperature obtained in the simulation. The anticipated temperature was approximately 550 Kelvin with the brake ducts installed. This expectation corresponds to the results obtained, which are subsequently discussed in Section 4.2.

4.1 MESH QUALITY

In order to ensure the accurate simulation of convection, an additional characteristic was measured from the results. y^+ is a dimensionless variable that describes the quality of the mesh. In critical regions, where the behavior near the walls necessitates precise modeling, the targeted value of y+ is less than 5. As can be seen in Figure 4.2, this limit was achieved through an iterative process involving a modification of the parameters of prismatic layers, as detailed in Section 3.2.2. [9]

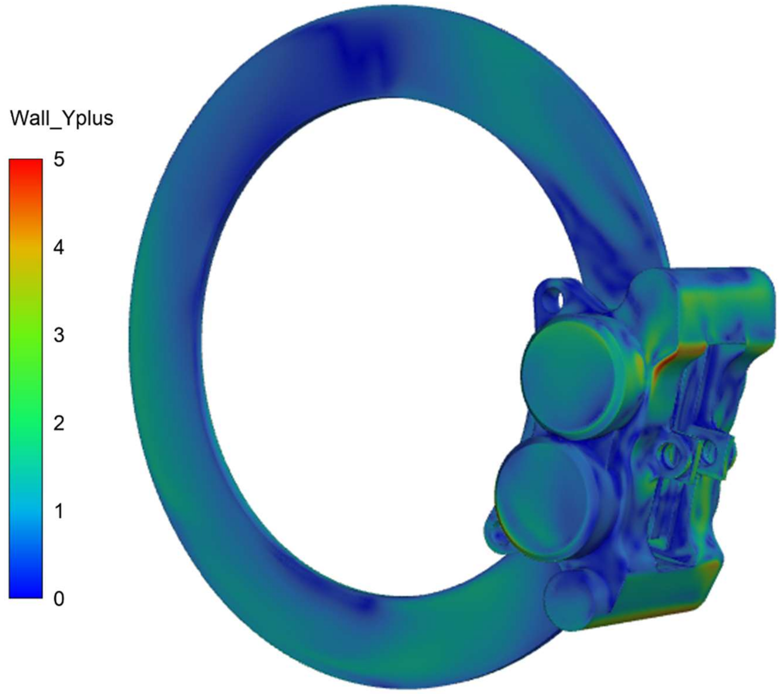

Figure 4.2: y^+ on the critical regions

4.2 COMPARISON OF THE DESIGNS

As stated in Section 3.1, the simulation was conducted using three distinct brake duct designs and one configuration that did not incorporate a brake duct. A comparative analysis of all the designs was conducted, with the temperature at a specific point on the surface of the calliper body and the average temperature of brake disc serving as the primary variables. As presented in Table 4.1, the implementation of the brake ducts is crucial to prevent the overheating of the brake system, a phenomenon that may result in brake fade and thermoelastic instability. Despite the demonstrated similarity in cooling capabilities among the designs, due to the hydraulic fluid wet boiling point of around 480 Kelvin being the most imminent cause of failure, Design B, which exhibited superior calliper cooling performance, was selected for implementation and further refinement. Therefore, next analysis will be conducted only on the comparison of the ductless design and the design B.

Table 4.1: Temperatures in the mesuring point with different duct designs

Design	without ducts	A	B	C
Caliper body temperature [K]	784	567	563	570
Averaged disc temperature [K]	1140	955	957	969

As depicted in Figure 4.3, the dissipation of heat into the caliper body was to a certain extent regulated by the air cavity and the titanium pistons, regardless of the external cooling implementation. The brake duct effectively facilitated a sufficient volume of air into the wheel hub, thereby achieving a cooling effect that decreased the temperature of the outer caliper by approximately 220 Kelvins. This cooling effect plays a crucial role in preventing the system from overheating. It is crucial to acknowledge that these temperatures represent extreme values that are hardly ever attained during the operation of the monopost.

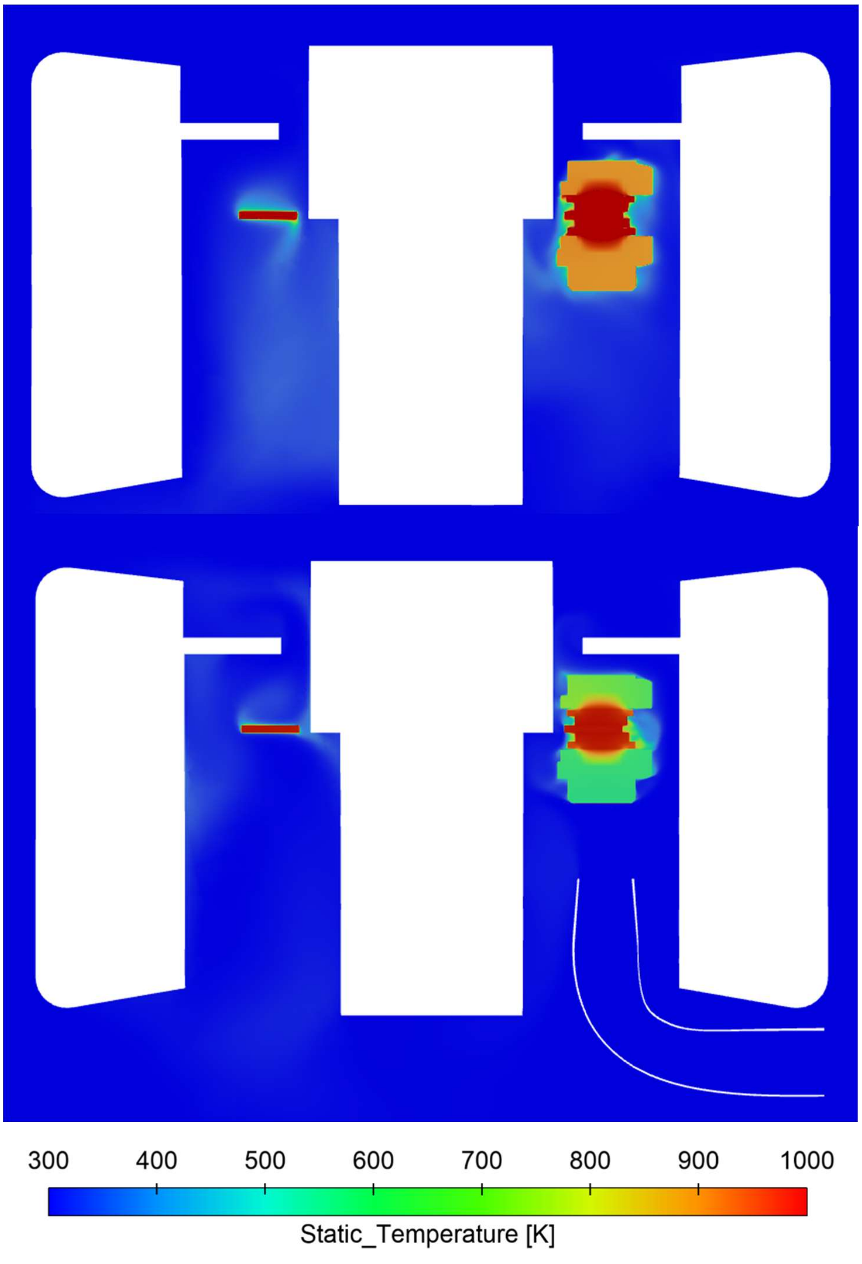

Figure 4.3: Comparison of temperature field contours without (top) and with (bottom) the brake duct

As evident in Figure 4.4, the CFD simulation results reveal a significant increase in airflow within the wheel assembly, a direct outcome of the optimized brake duct design, which enhances the convective cooling mechanism around the brake disc, as illustrated by the temperature distributions depicted in Figure 4.3. The effect is substantial, despite the flow influence from the front wing and rotating tires. The designed brake duct demonstrates successful maximization of the cooling effect by channeling airflow to the disc surface and reducing thermal gradients that contribute to failure modes such as brake fade and boiling of hydraulic fluid.

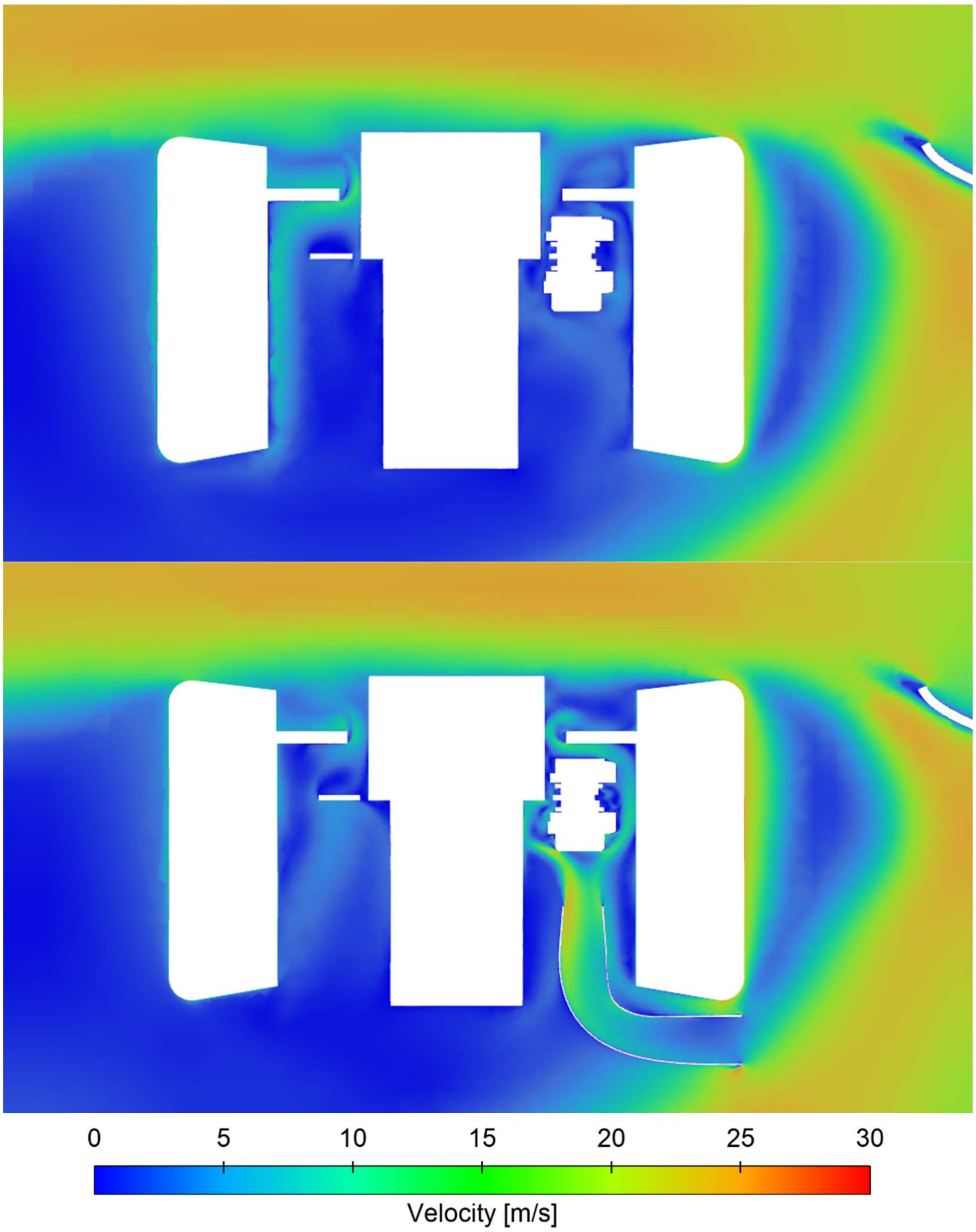

Figure 4.4: Comparison of air velocity magnitude field contours without (top) and with (bottom) the brake duct

CONCLUSION

The main objective of this thesis was to design a cooling system for a Formula Student monopost, perform a thermal analysis, and optimize the outlet of the cooling duct relative to the brake system.

This objective was achieved via comprehensive research on various brake systems and their performance with respect to cooling potential. From this analysis was determined that the optimal solution for this particular application is the implementation of in-house developed motorcycle disc brakes, in combination with convection cooling aided by the increased air flow by the brake ducts.

Three distinct designs of the brake ducts outlet were modeled. These three designs with the addition of a ductless configuration, were simulated using the setup process described in Section 3. The results of these simulations highlighted the importance of the brake duct, as the incorporation of any duct design resulted in a temperature reduction exceeding 200 Kelvins in comparison to the ductless configuration. Consequently, the brake duct that demonstrated the highest level of performance, as described in Section 4.2, will be manufactured and utilized in the ED5 monopost. It is imperative to note that the temperatures resulting from these simulations are extremes that are not ordinarily reached, even in the demanding operation conditions of the ED5 monopost.

Despite the consideration of highly improbable scenarios in simulations, the evident pressing nature of the hydraulic fluid boiling issue indicates that future development should prioritize the refinement of the brake duct design in conjunction with the implementation of torque vectoring and motor braking.

BIBLIOGRAPHY

[1] *Formula Student Germany International Design Competition*. Online. Formula Student Germany International Design Competition. Available at: https://www.formulastudent.de/about. [cit. 2025-02-26].

[2] *History of Formula SAE*. Online. Formula SAE. Available at: https://www.fsaeonline.com/page.aspx?pageid=c4c5195a-60c0-46aa-acbf-2958ef545b72. [cit. 2025-03-05].

[3] *TU Brno Racing*. Online. 2025. Available at: https://tubrnoracing.cz/. [cit. 2025-03-05].

[4] LIMPERT, Rudolf. *Brake Design and Safety (3rd Edition)*. 3. Warrendale: SAE International, 2011. ISBN 0768034388. Available at: https://doi.org/10.4271/R-398.

[5] *THE DESIGN OF FORMULA STUDENT BRAKE DISC*. BACHELOR'S THESIS. Brno: BRNO UNIVERSITY OF TECHNOLOGY, 2021.

[6] *What is Drum Brake? Working, Diagram, Construction and Applications*. Online. ElectricalWorkbook. 2025. Available at: https://electricalworkbook.com/drum-brake/. [cit. 2025-03-06].

[7] *Drilled vs slotted brake discs: which one to chose?* Online. Brembo. Available at: https://www.brembo.com/en/news-archive/disc-drilled-or-slotted. [cit. 2025-03-21].

[8] *Formula One Brake Systems, Explained!*. Online. AMG PETRONAS FORMULA ONE TEAM. 2025. Available at: https://www.mercedesamgf1.com/news/formula-one-brake-systems-explained. [cit. 2025-03-21].

[9] *CFD ANALYSIS OF FORMULA STUDENT IN TURN MOTION*. BACHELOR'S THESIS. Brno: BRNO UNIVERSITY OF TECHNOLOGY, 2023.

[10] *What are the Methods for Boundary Layer Mesh Generation — Lesson 2*. Online. Ansys Innovation Space. 2025. Available at: https://innovationspace.ansys.com/courses/courses/add-boundary-layers/lessons/what-are-the-methods-for-boundary-layer-mesh-generation-lesson-2/. [cit. 2025-04-17].

[11] *Fluent User's Guide*. Online. Ansys Help. 2025. Available at: https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v232/en/flu_ug/tgd_user_report.html%23tgd_user_report_qualitymeasure. [cit. 2025-04-17].

[12] *[CFD] Multiple Reference Frame (MRF) Approach for Turbomachinery*. Online. Available at: https://www.youtube.com/watch?v=oa-xcE0_0UY&t=4s&ab_channel=FluidMechanics101. [cit. 2025-05-17].

[13] *[CFD] Pseudo Transients for Steady-State CFD (Part 1) - Pseudo vs True Transients*. Online. Available at: https://www.youtube.com/watch?v=rF2t0-JmQZg&list=PLnJ8lIgfDbkon37Zbbh4wE_rp2PtX7aiR&index=8&ab_channel=FluidMechanics101. [cit. 2025-05-17].

[14] *Brake ducts*. Online. Constructorsf1.com. 2025. Available at: https://www.constructorsf1.com/en/clanky/brake-ducts-384.htm. [cit. 2025-03-21].

[15] *How Mercedes and Red Bull have diverged on F1 brake duct ideas*. Online. AUTOSPORT. 2025. Available at: https://www.autosport.com/f1/news/how-mercedes-and-red-bull-have-diverged-on-f1-brake-duct-ideas/6741111/. [cit. 2025-03-21].

[16] *What y+ should I use? Part 3 – Understanding impact of Y+ and number of prism layers on flow resolution*. Online. LEAP AUSTRALIA. 2025. Available at: https://www.leapaust.com.au/blog/cfd/y-plus_part3_understanding-impact-of-y-and-number-of-prism-layers-on-flow-resolution/. [cit. 2025-04-17].

LIST OF SYMBOLS AND ABBREVIATIONS

Symbol	**Term**	**Unit**
ρ	Density of the fluid	kg/m^3
c_p	Specific heat capacity of the fluid at constant pressure	$J/(kg \cdot K)$
T	Thermodynamic temperature	K
λ	Thermal conductivity of the fluid	$W/(m \cdot K)$
μ	Viscosity	$kg/(m \cdot s)$
P	Power output	W
A	Area	m^2
b	Thickness	m
T	Temperature	K
t	Time	s

LIST OF FIGURES

LIST OF TABLES