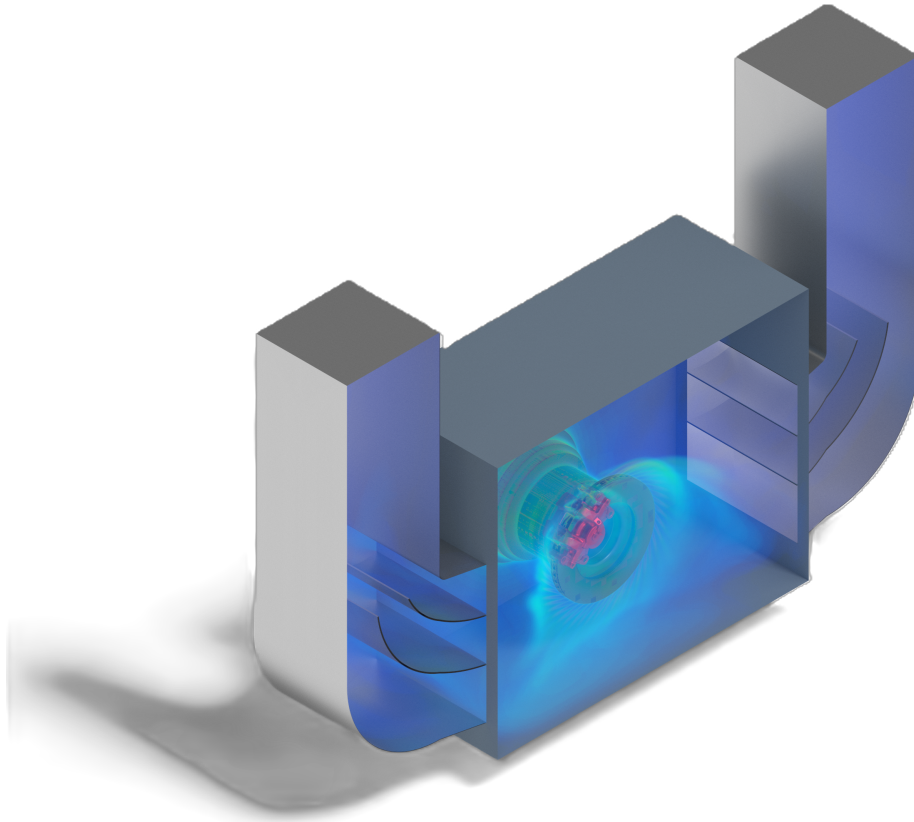




CHALMERS
UNIVERSITY OF TECHNOLOGY



Numerical Heat Transfer Simulations of Brake Discs under Dyno-Test Load Cases

Master's thesis in Mobility Engineering

VINOTH KANNA CHANDRASEKAR

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026
www.chalmers.se

MASTER'S THESIS IN MOBILITY ENGINEERING

Numerical Heat Transfer Simulations of Brake Discs under Dyno-Test Load Cases

VINOTH KANNA CHANDRASEKAR



CHALMERS
UNIVERSITY OF TECHNOLOGY

Department of Mechanics and Maritime Sciences
Division of Vehicle Engineering and Autonomous Systems
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026

Numerical Heat Transfer Simulations of Brake Discs under Dyno-Test Load Cases
VINOTH KANNA CHANDRASEKAR

© VINOTH KANNA CHANDRASEKAR, 2026.

Industrial Supervisor: Ilija, Todorovic, Volvo Cars Propulsion Components
Academic Supervisor and Examiner: Alexey Vdovin, Department of Mechanical Engineering

Master's Thesis 2026
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone +46 31 772 1000

Cover: Cross-section of the test-bench CFD model demonstrating the brake-caliper assembly. The color contour plot illustrates the air velocity distribution ranging from 0 m/s to over 20 m/s.

Printed by Chalmers Reproservice
Gothenburg, Sweden 2026

Numerical Heat Transfer Simulations of Brake Discs under Dyno-Test Load Cases

VINOTH KANNA CHANDRASEKAR

Department of Mechanics and Maritime Sciences

Division of Vehicle Engineering and Autonomous Systems

Chalmers University of Technology

Abstract

Automotive brake discs are critical safety components whose performance heavily relies on efficient thermal dissipation. Yet, conventional laboratory dynamometer testing remains costly and time-consuming. To address this challenge, this study establishes a reliable CFD methodology to simulate and visualize complex aero-thermal behavior before physical dynamometer testing, enabling evaluation even during the early design and development stages. Specifically, the thesis investigates the temperature-dependent Heat Transfer Coefficients (HTCs) of brake discs across a wide temperature and velocity spectrum.

The thermal simulations and analysis revealed a distinct decline in the localized HTCs of the brake disc surfaces as temperatures increased. By parameterizing the fluid boundaries, this work isolates the explicit influences of temperature-dependent fluid properties, rotor rotational speed, rotational direction, and the test-bench's volumetric inlet airflow rate on convective dissipation. This case study approach was systematically applied across both ventilated and solid brake disc geometries to evaluate structural cooling trade-offs. Finally, an experimental validation was conducted by mapping physical thermocouple data embedded within a test rotor against a transient thermal model in Abaqus FEM software, successfully correlating the numerical boundary conditions with real-world dynamometer temperature data.

Keywords: Disc brakes, CFD, heat transfer, dyno test, ventilated brake discs, solid brake discs, heat transfer coefficient, thermal analysis, FEM.

Preface

This report presents the outcome of my master’s thesis project carried out at the Department of Mechanics and Maritime Sciences at Chalmers University of Technology during the spring of 2026. This master’s thesis was carried out in collaboration with Volvo Cars Propulsion Components, with valuable technical support provided by Volvo Cars Corporation. The core objective of the project was to develop a numerical methodology and framework capable of replicating the thermal behavior of brake discs under various test procedures within a laboratory dynamometer environment. The master’s thesis project was supervised by Ilija Todorovic at Volvo Cars Propulsion Components Floby AB and examined by Alexey Vdovin at the Vehicle Engineering and Autonomous Systems (VEAS) Division at Chalmers University of Technology.

Acknowledgements

I want to extend my sincere gratitude to everyone who contributed, both directly and indirectly, to the successful completion of this master’s thesis project.

First and foremost, I am profoundly grateful to my academic supervisor and examiner, Alexey Vdovin from Chalmers University of Technology, and my industrial supervisor, Ilija Todorovic from Volvo Cars Propulsion Components, for their unwavering support, theoretical wisdom, and practical experience. I also appreciate the doctoral students within the VEAS Division for their invaluable advice and guidance during the examiner’s absence.

From Volvo Cars Corporation, I thank Jörgen Hansson, Mattias Eliasson, and their team for their technical support during the test-bench operations. Additionally, I am highly indebted to Volvo Cars Corporation for granting me access to their High-Performance Computing (HPC) cluster, which dramatically streamlined the numerical workload of this project.

I also convey my heartfelt gratitude to the entire VEAS Division for welcoming me into their academic community. Providing me with a dedicated workplace, access to necessary software licenses, and institutional resources was critical to executing this research successfully. Furthermore, I sincerely thank the division for the kind invitations to their internal events, which made my time during this thesis immensely enriching.

Vinoth Kanna Chandrasekar, Gothenburg, June 2026

Declaration on the use of ‘Artificial Intelligence’

I declare hereby that the usage of generative ‘Artificial Intelligence’ tools within the scope of this project was limited to basic numerical calculations, Python scripts for plotting graphs, summarizing research articles, LaTeX report suggestions or corrections, grammar check and improvements, and comprehension of some concepts. The thesis work was reviewed, and I take full responsibility for its contents.

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis, listed in alphabetical order:

BEVs	Battery Electric Vehicles
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
FEM	Finite Element Method
HTC	Heat Transfer Coefficient
MRF	Multiple Reference Frame
NVH	Noise, Vibration and Harshness
RANS	Reynolds Averaged Navier Stokes
RBM	Rigid Body Motion
RPM	Revolutions Per Minute
RSM	Reynolds Stress Model
SST	Shear Stress Transport
UN ECE	United Nations Economic Commission for Europe
VCPC	Volvo Cars Propulsion Components

Nomenclature

Below is the nomenclature of indices, sets, parameters, and variables that have been used throughout this thesis.

Indices

$cond$	Conduction
$conv$	Convection
$emit$	Emission
i, j	Index for direction of flow
ref	Reference
s	Surface
x, y, z	Index for coordinate direction

Parameters

L	Characteristic length in m
Nu	Nusselt number
Pr	Prandtl number
r_{dyn}	Dynamic rolling radius in m
Re	Reynolds number
α	Molecular diffusivity of momentum
δ_{ij}	Kronecker delta
Γ	Diffusivity (or) diffusion constant
σ	Stefan-Boltzmann constant in W/m ² K ⁴

Variables

A	Area in m ²
C_p	Specific heat in J/kg K
h	Heat Transfer Coefficient in W/m ² K
k	Thermal conductivity in W/m.K
k	Turbulent kinetic energy in m ² /s ²
$\dot{m}$	Mass flow rate in kg/s
N_{brake}	Brake rotation rate in revolutions per minute
$\dot{Q}$	Heat transfer in W
$\overline{S_{ij}}$	Mean Strain-Rate Tensor in 1/s
T	Temperature in °C
$\underline{u}$	Fluid velocity in m/s
u_τ	Shear velocity
v	Flow velocity in m/s
$\dot{V}$	Volume flow rate in m ³ /hr
v_{veh}	Vehicle speed in km/h
x	Distance in m
y	Distance from the wall in m
y^+	Non-dimensional distance from the wall
ϵ	Kinematic dissipation rate in m ² /s ²
μ	Dynamic viscosity in Pa.s
ν	Kinematic viscosity in m ² /s or molecular diffusivity of momentum
ν_T	Kinematic eddy viscosity or turbulent viscosity in m ² /s
ρ	Density in kg/m ³
τ_{ij}^{RS}	Reynolds stress tensor in m ² /s ²
ϕ	Flow variable
ω	Angular velocity in rad/s or specific dissipation rate in 1/s

Mathematical Operators

∇	Nabla operator
----------	----------------

Contents

List of Acronyms	xi
Nomenclature	xiii
Contents	xvii
List of Figures	xix
List of Tables	xxiii
1 Introduction	1
1.1 Aim and Objectives	1
1.2 Scope and Limitations	2
2 Theoretical Background	3
2.1 Automotive Brake Systems	3
2.1.1 Types of Automotive Brakes	4
2.1.2 Disc Brake Systems	5
2.1.3 Thermal Fading in Disc Brakes	6
2.2 Dynamometer Test-Bench	7
2.3 Governing Equations	8
2.3.1 Conservation of Mass (Continuity Equation)	8
2.3.2 Conservation of Momentum (Navier-Stokes Equation)	9
2.3.3 Conservation of Energy	9
2.4 Wall Functions in CFD	10
2.5 Turbulence Modeling	11
2.5.1 Reynolds Averaged Navier-Stokes (RANS)	12
2.6 Important Dimensionless Numbers	13
2.6.1 Reynolds Number	13
2.6.2 Prandtl Number	13
2.6.3 Nusselt Number	13
2.7 Rotation Modeling	14
2.7.1 Moving Wall	14
2.7.2 Multiple Reference Frame	14
2.8 Heat Transfer	15
2.9 CFD of a Disc Brake	16

3	Methodology	17
4	Pre-Processing	19
4.1	Geometrical Setup	19
4.2	Discretization	22
4.3	Physical and Boundary Models	23
4.3.1	Choice of Turbulence Model and Boundary Layer Study . . .	24
4.3.2	Motion Specification	26
4.3.3	Thermal Boundary Conditions	26
4.3.4	Domain Inlet and Outlet Conditions	27
4.4	Mesh Convergence Study	27
5	CFD Simulation Cases	31
5.1	Case Setup: Temperature-Dependent Fluid Properties	31
5.2	Case Setup: Baseline Configuration	32
5.3	Case Setup: Variable Rotational Speeds	33
5.4	Case Setup: Variable Rotational Direction	33
5.5	Case Setup: Variable Inlet Airflow Rate	33
5.6	Case Setup: Brake Design	34
6	Results and Discussion	37
6.1	Evaluation of HTC at Variable Fluid Properties	37
6.2	Evaluation of HTC of Baseline Configuration	38
6.3	Evaluation of HTC at Variable Rotational Speed	44
6.4	Evaluation of HTC at Variable Rotational Direction	47
6.5	Evaluation of HTC at Variable Inlet Airflow Rate	48
6.6	Evaluation of HTC for Variable Brake Design	50
7	Experimental Validation	53
8	Conclusions	57
8.1	Open Points	58
	Bibliography	59
A	Appendix 1	I
B	Appendix 2	V

List of Figures

2.1	Comparison of a brake disc to a drum brake [3]	4
2.2	Classification of brake discs based on various factors. Top left: classification based on caliper design [4], top right: classification based on disc design [5], bottom: classification based on ventilation design [6]	5
2.3	Idealized curve of temperature-dependent change in coefficient of friction of a brake disc [7]	6
2.4	Brake dynamometer [8]	7
2.5	Brake disc set-up inside the enclosure	8
2.6	Relation between dimensionless velocity and wall distance [10]	10
2.7	Schematic explanation of centrifugal action inside the ventilation cavity of a brake disc (The image was made using generative AI from a hand sketch)	16
3.1	Workflow followed for the study	17
4.1	CAD models of the rig. Left: CAD replica of the test-bench, right: cross-section to visualize inside the enclosure	19
4.2	3D Model of the chuck inside the enclosure	20
4.3	Brake discs used for this study. Left: ventilated brake disc 1, middle: ventilated brake disc 2, right: solid brake disc 3	20
4.4	Surface segmentation used for localized HTC mapping	21
4.5	Types of cells used for CFD. Left: tetrahedral cells, right: polyhedral cells [16]	22
4.6	Defined regions from the surfaces. Left: Fluid domain, right: rotational domain	24
4.7	Wall $y+$ plot against area of the brake surfaces	25
4.8	Computational mesh resolution within the fluid domain. Left: cross-sectional front view, right: magnified view of the prism layer grid resolving the near-wall thermal boundary layer inside the cooling vanes	25
4.9	Heat flux of ventilation cavity and drag coefficient against the number of cells	28
4.10	Averaged mass flow rate in the vanes against the number of cells	29
4.11	Cross sections of meshed computational domain. Left: front view, right: side view	29
5.1	Brake disc 2 and brake disc 3	35

6.1	Influence of fluid properties on surface-averaged HTC and heat transfer of the ventilation cavity (180 km/h, 300 m ³ /h)	38
6.2	Percentage deviation of the HTC values with variable properties as the baseline	38
6.3	Surface averaged HTC as a function of surface temperature (180 km/h, 300 m ³ /h)	39
6.4	Surface averaged heat transfer as a function of surface temperature (180 km/h, 300 m ³ /h)	39
6.5	Surface averaged heat flux as a function of surface temperature (180 km/h, 300 m ³ /h)	39
6.6	Dimensionless numbers against temperature. Left: Reynolds and Prandtl number, right: Nusselt number	40
6.7	Cross sections of the computational Domain with the baseline configuration (500 °C, 180 km/h, 300 m ³ /h)	41
6.8	Heat transfer distribution (180 km/h, 300 m ³ /h)	42
6.9	HTC of Outer Hub Interface (500 °C, 180 km/h, 300 m ³ /h)	43
6.10	Averaged mass flow rate in the ventilation cavity (180 km/h, 300 m ³ /h)	43
6.11	Surface averaged HTC as a function of surface temperature at 30 km/h and 180 km/h (500 °C, 300 m ³ /h)	44
6.12	Averaged flow fields for 30 km/h and 180 km/h (500 °C, 300 m ³ /h)	45
6.13	Surface averaged HTC as a function of rotational speeds at 100°C, 300°C and 500°C (180 km/h, 300 m ³ /h)	46
6.14	Surface plot for averaged HTC of ventilation cavity as a function of rotational speed and temperature	46
6.15	Surface averaged HTC as a function of rotational speed at clockwise (CW) and counterclockwise (CCW) rotational direction (500 °C, 180 km/h, 300 m ³ /h)	47
6.16	Velocity plot for clockwise (CW) and counterclockwise (CCW) rotation (500 °C, 180 km/h, 300 m ³ /h). Top left: x-plane CW, top right: x-plane CCW, bottom left: z-plane CW, bottom right: z-plane CCW	48
6.17	Surface averaged HTC as a function of test-bench inlet airflow rates (500 °C, 180 km/h)	49
6.18	Velocity plot for different test-bench inlet airflow rates (500 °C, 180 km/h). Top: 300 m ³ /h, bottom: 1200 m ³ /h)	49
6.19	Comparison of HTC of friction and radial surface for brake disc 2 vs brake disc 2 with closed vents vs solid brake disc	50
6.20	Comparison of Heat transfer of friction and radial surface for brake disc 2 vs brake disc 2 with closed vents vs solid brake disc	50
6.21	Comparison of velocity at z-plane for brake disc 2 with open and closed vanes	51
6.22	Comparison of Heat distribution by the surfaces of the ventilated brake disc (Brake disc 2) and solid brake disc (Brake disc 3)	52
7.1	Schematic visualization of the location of the thermocouple on a ventilated brake disc indicated by the red dot	53
7.2	Speed of a single braking cycle from one of the test cases	53

7.3	Deceleration of a single braking cycle from one of the test cases . . .	54
7.4	Symmetric sliced part of the brake disc used for FEM simulations . .	54
7.5	Heat flux curve for mechanical braking	55
7.6	Sequence stop test with temperature from dyno and CFD/FEM simulations	55
B.1	Surface averaged heat transfer as a function of rotational speed and temperature ($300\text{ m}^3/\text{h}$)	V
B.2	Heat transfer distribution across the speed spectrum ($500\text{ }^\circ\text{C}$, $300\text{ m}^3/\text{h}$)	V
B.3	Surface response plots for segmented surface using the generalized equation 6.1	VI
B.4	Surface averaged heat transfer as a function of rotational speed and direction ($500\text{ }^\circ\text{C}$, $300\text{ m}^3/\text{h}$)	VII
B.5	Surface averaged heat transfer as a function of test-bench inlet airflow rate and temperature (180 km/h)	VII
B.6	Heat transfer distribution across the test-bench inlet airflow rate spectrum ($500\text{ }^\circ\text{C}$, 180 km/h)	VIII

List of Tables

2.1	Specifications in the general conservation equation	10
4.1	Geometrical overview of the brake discs	21
4.2	Physics solvers and models enabled	23
4.3	Prism Layer base Parameters	25
4.4	Mesh Parameters	26
4.5	Boundary and initial conditions used in the simulation	27
4.6	Mesh convergence cases	28
5.1	Case setup and boundary conditions for the fluid properties	32
5.2	Case setup for influence of rotational speed on heat transfer	33
5.3	Case setup for influence of volume flow rate speed on heat transfer	34
5.4	Case setup for influence of ventilation cavity on heat transfer	35
A.1	Surface-averaged HTC values ($\text{W}/\text{m}^2\text{K}$) for the baseline configuration (Brake disc 1) at various temperatures ($^{\circ}\text{C}$) (180 km/h , $300 \text{ m}^3/\text{h}$)	I
A.2	Surface-averaged heat flux values (W/m^2) for the baseline configuration (Brake disc 1) at various temperatures ($^{\circ}\text{C}$) (180 km/h , $300 \text{ m}^3/\text{h}$)	II
A.3	Surface-averaged heat transfer values (W) for the baseline configuration (brake disc 1) for surface temperatures ($^{\circ}\text{C}$) (180 km/h , $300 \text{ m}^3/\text{h}$)	II
A.4	Surface-averaged HTC values (W) for the baseline configuration (Brake disc 1) for surface temperatures ($^{\circ}\text{C}$) at 30km/h and 180 km/h ($300 \text{ m}^3/\text{h}$)	III
A.5	Coefficients of generalized HTC equation for brake disc 1 ($300 \text{ m}^3/\text{h}$)	III
A.6	Surface-averaged HTC values (W) for the baseline configuration (Brake disc 1) at various temperatures ($^{\circ}\text{C}$) at 30km/h and 180 km/h ($500 \text{ }^{\circ}\text{C}$, $300 \text{ m}^3/\text{h}$)	IV
A.7	Coefficients of generalized HTC equation for brake disc 2 ($300 \text{ m}^3/\text{h}$)	IV

1

Introduction

Since their invention in the early 20th century, braking systems have remained a critical component of mobility infrastructure, ranging from passenger vehicles to heavy-duty freight trains. The primary function of a brake system is to decelerate a moving body efficiently by converting kinetic energy to thermal energy through friction, potential energy to thermal energy (downhill driving), or to maintain the vehicle stationary.

Despite the rise of BEVs and regenerative braking, which recaptures energy during deceleration, mechanical friction brakes remain indispensable for emergency maneuvers, high-rate deceleration, and parking. Because a braking system's reliability is non-negotiable, it is subjected to various degradation factors that can compromise performance. Consequently, rigorous testing is mandatory to ensure compliance with stringent safety standards.

However, physical prototyping and experimental testing are often cost-prohibitive and time-intensive. To streamline the development cycle, high-fidelity CFD models offer a reliable alternative, providing results that closely approximate physical experimental data. Utilizing CFD allows for rapid design iterations and performance validation without the need for constant physical prototyping. This research focuses on the development of a robust CFD model designed to simulate standardized test conditions, to monitor their thermal behavior under such test conditions, and to produce validated results for direct comparison with experimental benchmarks.

1.1 Aim and Objectives

The primary focus of this thesis is to establish a robust numerical methodology for computing temperature-dependent Heat Transfer Coefficients (HTC) and to conduct a sensitivity analysis of an automotive brake disc using Computational Fluid Dynamics (CFD). Furthermore, the derived HTC values are integrated into a transient thermal FEM model to predict the rise in temperature and heat dissipation characteristics during various braking sequences.

To achieve this aim, the research addresses the following specific objectives:

- **Characterization of HTC:** Determination of localized HTC values across the diverse surfaces of an automotive gray cast iron brake disc.
- **Sensitivity Study:** Evaluation of HTC sensitivity across multiple variables, including:

1. **Geometric Configurations:** A comparative analysis between solid and ventilated disc designs.
2. **Aerodynamic Influence:** Qualitative and quantitative assessment of heat transfer and corresponding metrics based on the interaction of the brake disc with the air and the dependency on cooling airflow rates supplied to the test-bench.
3. **Operational Load Cases:** Evaluation based on standardized test sequences within a controlled dynamometer environment.

1.2 Scope and Limitations

Given the 20-week duration allocated for this Master's thesis, specific simplifications, assumptions, and numerical boundaries have been established to maintain a focused and high-quality analysis. The scope for the research is governed by the following constraints:

- **System boundaries:** The computational domain is entirely confined to the brake disk and caliper assembly. Other components of the braking system, such as hydraulic actuators, brake boosters, or electronic control units, are excluded from this study.
- **Time Domain:** Since the primary objective of this study is to characterize temperature-dependent local HTC's, a steady-state framework is implemented, and dependency of time of any kind (solvers, rotation modeling) is neglected.
- **Environmental Domain:** The CFD simulations for HTC's are designed specifically to replicate brake dynamometer (dyno) test conditions. Certain components of the test-bench are simplified. For example, the mounting fixtures are omitted from the fluid domain to conserve computational resources.
- **Heat Transfer Mechanism:** Thermal evaluation is focused exclusively on convective heat transfer. Thermal radiation and solid-body thermal conduction are excluded from the computation.
- **Material Properties:** Due to the exclusion of conduction and radiation, it is not necessary to impose material properties for the solid brake disc. Instead, the disc geometries are treated as zero-thickness boundary surfaces, which provides a highly consistent and computationally efficient baseline for convective HTC characterization.

2

Theoretical Background

The objective of this section is to establish the fundamental theoretical framework required to interpret and support the subsequent numerical analysis. It provides an overview of automotive braking systems and the governing principles of fluid mechanics.

2.1 Automotive Brake Systems

Understanding the fundamental functions of these systems is essential for contextualizing the sensitivity study and the physical parameters derived from the numerical simulations. The primary objectives of a braking system include:

- decelerating the vehicle from a specific velocity to a complete standstill
- applying a holding force to maintain the vehicle's position when stationary
- ensuring consistent performance across a wide range of thermal and environmental conditions and
- adhering to stringent legal and safety requirements.

Automotive braking systems are categorized based on their functional and specific operational use cases: The primary braking system (Service brake) acts as the main system during standard operation to modulate vehicle speed. The secondary braking system acts as a dormant circuit to provide deceleration in the event of a primary system failure. An emergency brake system is designed to mitigate impact or stabilize the vehicle during critical safety incidents. And finally, a parking brake is a mechanical or electromechanical system designed to hold the vehicle stationary, particularly on inclined surfaces. [1]

The mentioned braking systems should fulfill the criteria of certain legislative requirements before receiving approval for introducing the automobile to the market. For instance, according to UN ECE regulations, a passenger car must meet the following specific performance benchmarks:

- With the engine engaged, the **primary braking system** must achieve a minimum deceleration of 6.43 m/s^2 with a maximum pedal force of 500 N.
- The **secondary braking system** must maintain a minimum deceleration capability of 2.44 m/s^2 upon primary failure.
- The **emergency braking** setup must provide a deceleration of at least 5 m/s^2 during emergency interventions

- The **parking brake** must hold the vehicle stationary on a gradient of 18% without slipping [2]

2.1.1 Types of Automotive Brakes

Within the automotive domain, mechanical friction braking systems remain the industry standard, despite the diversification of electromagnetic, hydraulic, and pneumatic (air) brakes, and are primarily categorized into drum brakes and disc brakes. The selection between these two architectures (Figure 2.1) is governed by their distinct mechanical characteristics. Drum brakes feature an enclosed design that resists environmental contamination and reduces manufacturing costs, while benefiting from a mechanical 'self-energizing' effect that amplifies clamping forces. However, they are highly temperature-sensitive. Thermal expansion causes the drum to expand away from the linings, resulting in improper contact and increased brake pedal travel.

Conversely, disc brakes utilize an open architecture that provides superior convective cooling and exceptional resistance to thermal brake fade. Lacking aggressive self-energization, they maintain consistent stability against fluctuations in the friction coefficient (μ). Additionally, disc brakes possess an inherent ability to self-adjust as wear occurs and offer a simpler replacement procedure for friction components. Due to this superior thermal performance and stability under heavy loads, disc brakes have become the standard for the front axles of passenger cars and heavy vehicles, while drum brakes are predominantly utilized in rear-wheel applications or parking brake systems [1].

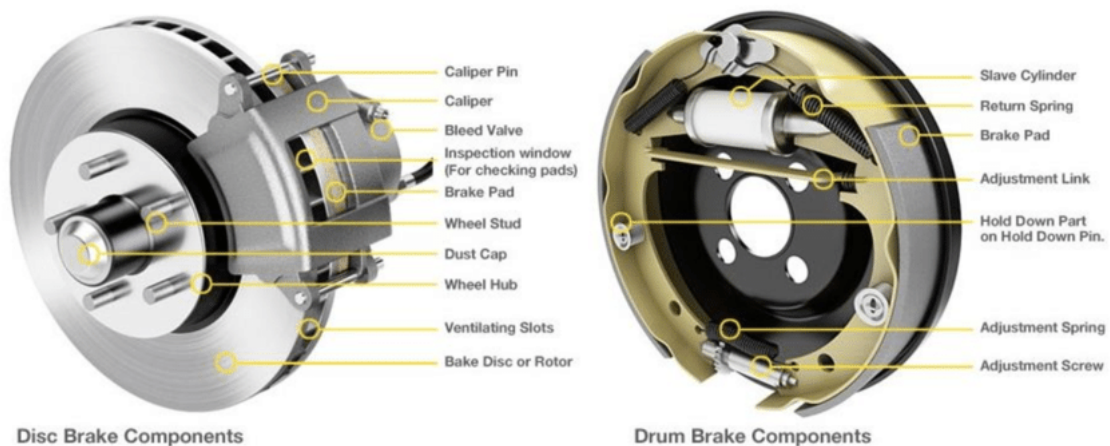


Figure 2.1: Comparison of a brake disc to a drum brake [3]

2.1.2 Disc Brake Systems

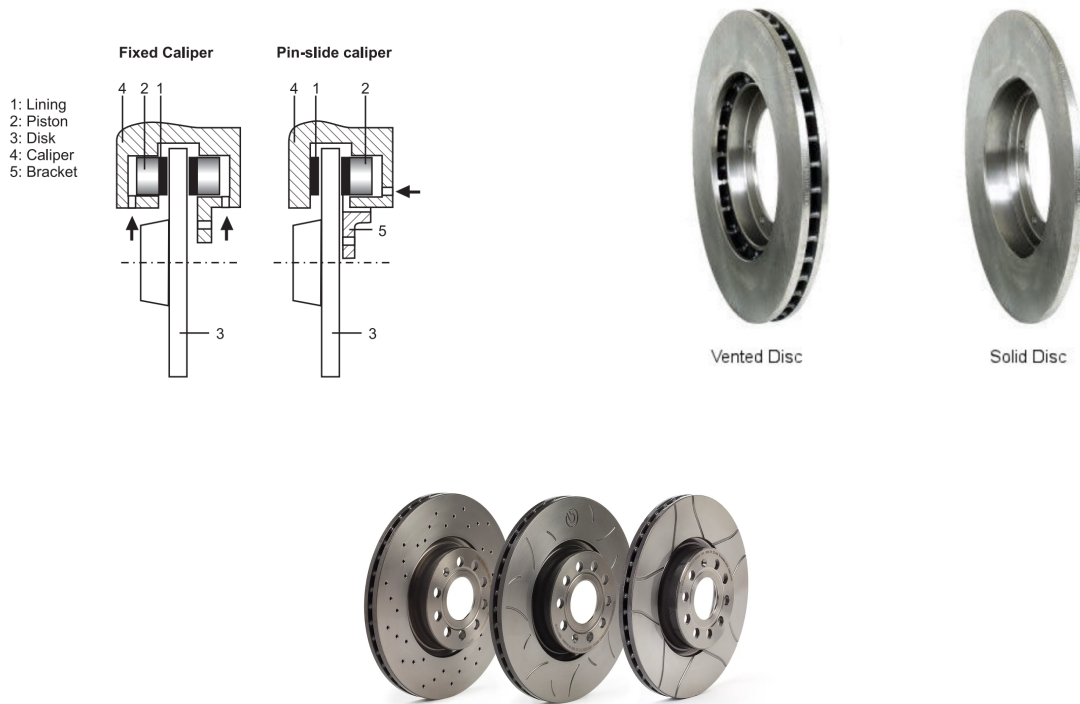


Figure 2.2: Classification of brake discs based on various factors. Top left: classification based on caliper design [4], top right: classification based on disc design [5], bottom: classification based on ventilation design [6]

The fundamental components of a disc brake assembly are illustrated in Figure 2.2. The rotor is coupled to the wheel hub, rotating synchronously with the vehicle's tires. In contrast, the caliper is mounted to the vehicle's suspension, rendering it stationary relative to the rotating disc. The caliper assembly houses the hydraulic pistons and friction pads required to exert clamping pressure. Disc brakes are generally classified based on their actuation method, rotor geometry, and caliper architecture.

- **Disc brake types based on caliper design:** Disc brakes are categorized into floating (sliding) or fixed calipers (Figure 2.2 top left). Floating calipers utilize a single-sided piston arrangement, sliding on pins to engage both pads. Conversely, fixed calipers employ pistons on both sides of the rotor, offering a more rigid structure and uniform pressure distribution.
- **Disc brake types based on rotor (disc) design:** Solid discs feature a thinner profile, lower mass, and reduced manufacturing costs, but lack internal ventilation, making them susceptible to thermal deformation. In contrast, ventilated discs incorporate internal vanes that pump airflow through the core for superior convective cooling, though this architecture requires a wider physical space and increases component weight. (Figure 2.2 top right)

- **Disc brake types based on cooling design:** Beyond internal vanes, surface modifications like slots and drilled holes are utilized (Figure 2.2 bottom) to stabilize the friction coefficient (μ) by providing paths for contaminants. Additionally, these features facilitate the expulsion of gases generated by the 'out-gassing' of brake pad resins at high temperatures, which could otherwise form an insulating film and degrade braking performance [6].

2.1.3 Thermal Fading in Disc Brakes

As discussed in the section 2.1, the braking system must be reliable at all points during the operation. Hence, there are high standard constraints with respect to:

- mechanical aspects like structural integrity and fatigue life, etc.
- thermal aspects like thermal distribution, etc.
- NVH aspects like high-frequency acoustic and structural disturbances during deceleration.

This study focuses exclusively on the thermal behavior of brake discs. When analyzing the thermal characteristics of a braking system, "fading" is a critical phenomenon that directly impacts operational reliability. This thermal condition is illustrated in Figure 2.3, which plots the relationship between the brake disc temperature and the friction coefficient (μ). By examining this relationship, one can identify the thermally stable and thermally critical operating ranges of the brake system. Fading occurs when the brakes are subjected to high thermal loads, typically resulting from frequent applications or high-rate decelerations.

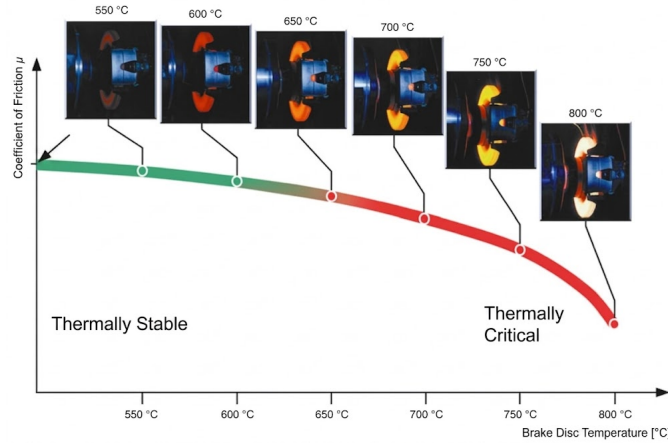


Figure 2.3: Idealized curve of temperature-dependent change in coefficient of friction of a brake disc [7]

The phenomenon is characterized by a significant reduction in the friction coefficient as temperature increases. This degradation in friction performance results in a longer brake pedal travel, thereby creating an unsafe operating environment [7]. To ensure vehicle safety, it is imperative to implement design measures, such as optimized cooling geometries, that keep the disc within the thermally stable region of the performance plot.

2.2 Dynamometer Test-Bench

This study utilized the brake dynamometer test-bench facility at Volvo Cars Torslanda in Gothenburg. Generally, a dynamometer is a testing equipment used to measure the rotational inertia, torque, or power of a Device Under Test (DUT), such as an engine or electric motor, under certain working conditions. In brake testing, a dynamometer serves a similar purpose, allowing engineers to evaluate mechanical parameters (such as stress and strain), thermal properties (including cooling efficiency and radiation effects), or to simulate real-world driving cycles to verify that the braking system conforms to legislative regulations.

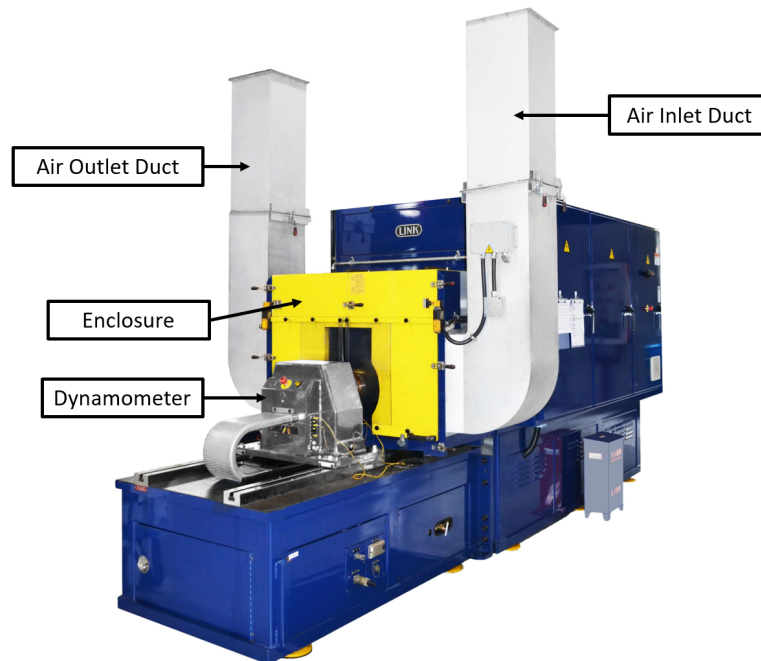


Figure 2.4: Brake dynamometer [8]

The brake dynamometer configuration similar to the one used in this project is shown in Figure 2.4. The white duct on the right functions as the inlet that supplies regulated cooling air to the test-bench, while the white duct on the left serves as the exhaust outlet. The yellow component represents the structural enclosure; brake testing must be performed in a closed environment for safety reasons to isolate high-energy rotating components. Additionally, because friction braking generates significant amounts of wear particles (brake dust), the enclosure confines these emissions to prevent them from contaminating the surrounding facility. The mounting chuck, shown in Figure 2.5, that holds the brake disc is positioned inside this enclosure. Adjacent to it, the gray housing encapsulates the electric motor responsible for generating the driving torque or measuring the counteracting braking forces. Finally, a rigid fixture framework is mounted to the test-bench next to the dynamometer to securely position the brake caliper assembly and integrate instrumentation units, such as thermocouples.

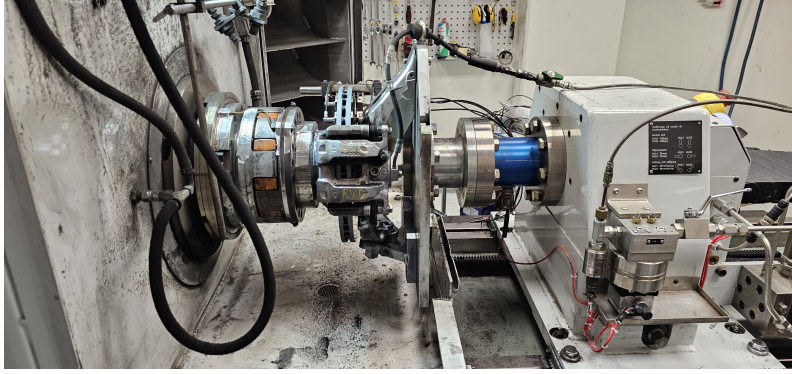


Figure 2.5: Brake disc set-up inside the enclosure

2.3 Governing Equations

Fluid flow simulations are governed by the fundamental conservation laws of physics: the conservation of mass, momentum, and energy. These principles are mathematically expressed through a set of partial differential equations, which form the essential framework for analyzing and predicting fluid behavior.

These equations ensure that the total quantity of a physical property remains conserved within a closed system or is accurately accounted for as it enters or leaves a control volume. The basic form of the governing equations is given by:

$$\frac{\partial}{\partial t}(\rho\phi) + \underbrace{\nabla \cdot (\rho u)\phi}_{F_\phi} = \underbrace{\nabla \cdot (\Gamma \nabla \phi)}_{D_\phi} + Q_\phi \quad (2.1)$$

In the above general transport equation, the variable ϕ represents a flow variable, such as the flow velocity u or specific enthalpy h . The first term of the equation represents the unsteady term, the second term denotes the convective flux (F_ϕ), which is governed by the flow of the quantity ϕ . On the other side of the equation, the diffusive flux (D_ϕ) represents the transport driven by spatial gradients in the quantity ϕ , and the final term represents the source from within the system Q_ϕ . By the definition of the flow quantity and the inclusion or exclusion of specific terms, this general transport equation can be adapted to formulate the specific conservation laws for mass, momentum, and energy. [9]

2.3.1 Conservation of Mass (Continuity Equation)

The mass flow into a control volume or a system, as defined by classical mechanics, should either come out of the system or increase the net mass of the system. The continuity equation is defined mathematically as

$$\dot{m} = \rho \cdot A \cdot v \quad , \text{ where } A \text{ is the area and } v \text{ is the velocity} \quad (2.2)$$

By substituting the flow variable for ϕ with a constant 1 and setting the diffusive and source terms in (1) to zero, the continuity equation is mathematically expressed as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \cdot u)}{\partial x} + \frac{\partial (\rho \cdot v)}{\partial z} + \frac{\partial (\rho \cdot w)}{\partial y} = 0 \quad (2.3)$$

and in coordinate-free notation, it can be written as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \cdot \vec{v}) = 0 \text{ or for incompressible flows } \nabla \cdot \vec{v} = 0.$$

2.3.2 Conservation of Momentum (Navier-Stokes Equation)

When the conservation of momentum principle is applied to a fluid continuum, often referred to as the Navier-Stokes equations, it dictates that the time rate of change of momentum within a control volume, combined with the net momentum flux across its boundaries, must equal the sum of all external forces acting on the fluid. The Navier-Stokes equation takes into account the interaction of various forces that influence the momentum balance.

For conservation of momentum, the flow variable in the general equation is set to the flow velocity vector u_i , where i is the vector notation for x , y , and z directions, the diffusion term to the viscous forces, and the source term to the body forces, which can mathematically be expressed for an incompressible Newtonian fluid as:

$$\underbrace{\rho \left(\frac{\partial u_i}{\partial t} + (u_i \cdot \nabla) u_i \right)}_{\text{acceleration forces (inertia)}} = \underbrace{F_z}_{\text{volume forces}} - \underbrace{\nabla p}_{\text{pressure forces}} + \underbrace{\mu \nabla^2 u_i}_{\text{shear stress forces (viscous forces)}} \quad (2.4)$$

2.3.3 Conservation of Energy

As discussed before, the thesis project involves the heat transfer coefficient, which means there is also a need to take the conservation of energy into consideration. It is obtained by replacing the flow variable with temperature, dissipation with Fourier conduction, and source as a collection of pressure work, heat source, and viscous dissipation, resulting in the equation for conservation of energy.

$$\underbrace{\rho c_p \frac{\partial T}{\partial t}}_{\text{Transient thermal energy storage}} + \underbrace{\rho c_p (u \cdot \nabla T)}_{\text{Convective heat transport}} = \underbrace{\left(\frac{\partial p}{\partial t} + u_i \cdot \nabla p \right)}_{\text{Pressure work}} + \underbrace{\nabla \cdot (\lambda \nabla T)}_{\text{Thermal conduction}} + \underbrace{\rho \dot{q}_s}_{\text{Heat source}} + \underbrace{\mu \Phi}_{\text{viscous dissipation}}$$

Ultimately, the three conservation equations can be summarized in the form of a table as follows:

Table 2.1: Specifications in the general conservation equation

Conservation of	ϕ	D_ϕ	Q_ϕ
Mass	1	0	0
Momentum	u_i	$\mu \nabla^2 u_i$	$-\nabla p + F_z$
Energy	T	$\nabla \cdot (\lambda \nabla T)$	$(\frac{\partial p}{\partial t} + u_i \cdot \nabla p) + \rho q_s + \mu \Phi$

2.4 Wall Functions in CFD

To accurately simulate a turbulent boundary layer, it is essential to define the y^+ value, also referred to as the dimensionless wall distance. The y^+ value serves as a scaling parameter to determine the appropriate mesh density and turbulence modeling approach near solid boundaries. It is defined as the ratio of the product of the distance from the wall to the first cell centroid (y) and the shear velocity (u_τ) to the kinematic viscosity (ν):

$$y^+ = \frac{y u_\tau}{\nu} \quad (2.5)$$

In heat transfer studies, capturing near-wall physics is vital, as the primary thermal interaction is concentrated within this region. The relationship between y^+ and the dimensionless velocity in Figure 2.6 illustrates the division of the turbulent boundary layer into three distinct sub-regions, driven by the transport of momentum from the free stream toward the wall via turbulent eddies:

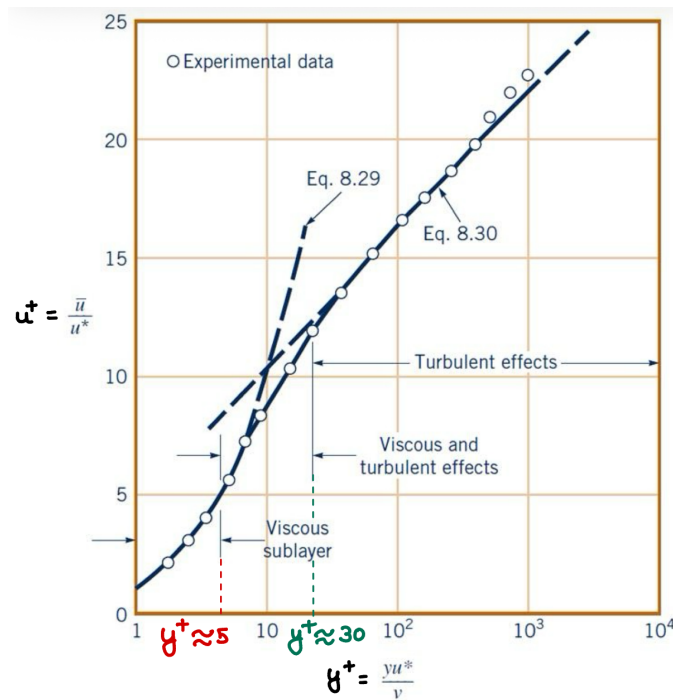


Figure 2.6: Relation between dimensionless velocity and wall distance [10]

- **Viscous Sublayer** ($0 < y^+ < 5$): In the immediate vicinity of the wall, viscous forces dominate, rendering the flow locally laminar. Here, the velocity profile is linear ($u^+ = y^+$).
- **Buffer Layer** ($5 < y^+ < 30$): This represents a transition zone where neither viscous nor turbulent shear stresses dominate. Because of its mathematical complexity, this region cannot be easily represented by a single analytical law.
- **Log-law Region** ($y^+ > 30$): Dominated by turbulent mixing, momentum transfer in this zone is primarily due to turbulent eddies, and the velocity profile follows a logarithmic distribution.

In CFD practice, it is generally recommended to avoid placing the first cell centroid within the buffer layer, which gives rise to two conventional wall modeling approaches:

- **Wall resolved or low- y^+ approach:** The steep velocity gradients and shear stress variations within the viscous sublayer are resolved directly. This is highly suitable for separated flows. The centroid of the first cell must be placed very close to the wall, typically $y^+ \leq 1$. It is standard practice to use a minimum of 10 prism layers to capture the transition into the turbulent core adequately.
- **Wall modeled or high- y^+ approach:** This method approximates near-wall behavior using semi-empirical wall functions. It is computationally efficient and suitable for attached flows without strong adverse pressure gradients. The first cell centroid is placed within the log-law region ($30 < y^+ < 300$), often requiring only 2–4 prism layers to bridge the wall and the outer flow.

Blended approach: Blending functions are used in modern CFD applications to overcome the nonphysical behavior of the first cell centroid in the buffer region.

2.5 Turbulence Modeling

Almost all of the technical and real-world flows are predominantly turbulent in nature. This turbulent regime triggers stochastic, three-dimensional fluctuations that drastically enhance momentum and thermal energy exchange. Because the chaotic nature of turbulence precludes exact analytical predictions for complex geometries, specialized turbulence models are employed to approximate these fields and their impact on momentum and heat transfer.

The primary objective of turbulence modeling is to account for the influence of turbulent fluctuations on the mean flow field. In computational fluid dynamics, these fields are mathematically approximated using Reynolds decomposition, where an instantaneous property ϕ is split into a time-averaged mean component and a fluctuating component: $\phi = \bar{\phi} + \phi'$.

Selecting a specific turbulence model depends largely on the complexity of the flow and the degree of separation to be resolved. The most common computational modeling approaches are direct numerical simulation, large eddy simulation (and

its variants), detached eddy simulation (and its variants), and Reynolds Averaged Navier-Stokes. Because DNS, LES, and DES models require resolving transient, small-scale turbulent structures that demand prohibitive computational resources, they were not utilized for this thesis project and are excluded from the discussion.

2.5.1 Reynolds Averaged Navier-Stokes (RANS)

In this method, the Navier-Stokes equations are time-averaged using Reynolds decomposition. Applying this process to the governing equation (4) yields:

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + [(\bar{u}_i \cdot \nabla) \bar{u}_i + (u'_i \cdot \nabla) u'_i] \right) = \bar{F}_z - \nabla \bar{p} + \mu \nabla^2 \bar{u}_i + \rho \nabla \cdot \overline{u'_i u'_j} \quad (2.6)$$

Due to the non-linearity of the original equations, time-averaging introduces an unknown term, the Reynolds stress tensor, $-\rho \overline{u'_i u'_j}$, and the equation becomes open. This term represents the influence of turbulent fluctuations on the mean flow. To close the system of equations, the tensor containing the velocity components must be approximated through turbulence modeling. The two primary modeling strategies are the Reynolds Stress Model (RSM) and the Eddy Viscosity Model (based on the Boussinesq Hypothesis). The Reynolds Stress Model is not discussed here due to its irrelevance to this project.

Eddy Viscosity Model

The Eddy Viscosity models are based on the Boussinesq hypothesis, which assumes that Reynolds stress is proportional to the mean strain rate. This relationship is approximated as:

$$\tau_{ij}^{RS} = \nu_T 2 \overline{S_{ij}} - \frac{2}{3} k \delta_{ij}$$

where the eddy viscosity ν_T acts as the proportionality factor. In this way, the number of unknowns is reduced from 6 to 2, the turbulent kinetic energy and eddy viscosity. Based on these two unknowns as the basis, they are classified as one-equation and two-equation turbulence models.

- **Spalart-Allmaras (1-Equation):** A computationally efficient model developed for aerodynamic flows. While it excels in computing boundary layers for external aerodynamics, it often under-predicts separation in flows with sharp gradients or decaying turbulence.
- **$k - \epsilon$ (2-Equation):** A high- Re model that relies on wall functions ($y^+ > 30$). It is robust for freestream regions but struggles with adverse pressure gradients and strong curvatures. The Realizable $k - \epsilon$ variant improves on this by using a variable C_μ to prevent non-physical behavior.
- **$k - \omega$ (2-Equation):** A low- Re model used to resolve the viscous sub-layer ($y^+ \approx 1$). It is superior for sheared flows and adverse pressure gradients but can be sensitive to freestream boundary conditions.

- **$k-\omega$ SST Menter:** The Shear Stress Transport (SST) model uses a blending function to combine the strengths of both: $k-\omega$ in the near-wall region and $k-\epsilon$ in the freestream. It is widely considered the most accurate RANS model for predicting flow separation, though it may slightly over-predict turbulence in regions of strong acceleration or stagnation. [9]

2.6 Important Dimensionless Numbers

To characterize the fluid flow and qualitatively evaluate both the aerodynamic and thermal performance profiles, a few dimensionless governing parameters are introduced. Because these dimensionless numbers are utilized later in this project to analyze and interpret the simulation data, a brief theoretical introduction to each parameter is presented in this section.

2.6.1 Reynolds Number

In external flow over a surface, viscous forces and the "no-slip" condition create an aerodynamic boundary layer. The flow regime within this layer is quantified by the Reynolds number, given by

$$Re = \frac{\rho U L}{\mu} \quad (2.7)$$

which represents the ratio of inertial forces to viscous forces. At low Re , the flow is laminar, exhibiting smooth layers and minimal transversal mixing. As velocity (U) or characteristic length (L) increases, inertial forces dominate, causing a transition to turbulent flow.

2.6.2 Prandtl Number

The Prandtl number relates the velocity boundary layer to the thermal boundary layer by defining the ratio of the molecular diffusivity of momentum (ν) to the molecular diffusivity of heat (α). It is the measure of whether momentum or heat spreads faster in a fluid and is expressed as:

$$Pr = \frac{\nu}{\alpha} = \frac{C_p \mu}{k} \quad (2.8)$$

A low Prandtl number ($Pr < 1$) characterizes fluids such as gases and liquid metals where heat diffuses faster than momentum. Conversely, a high Prandtl number ($Pr > 1$) applies to the viscous fluids like oils where momentum diffusion dominates.

2.6.3 Nusselt Number

The Nusselt number represents the dimensionless temperature gradient at a surface and is essentially a dimensionless HTC. It is derived from correlations between the

Reynolds number and the Prandtl number. The Nusselt number is given by the relation

$$Nu = \frac{hL}{k} = CRe^m Pr^n \quad (2.9)$$

where L is the characteristic length, C the geometry-dependent constant, and m and n are empirical exponents defined based on the flow regime and flow properties. [11]

2.7 Rotation Modeling

To replicate real-world operating conditions, the rotation of the brake rotor must be accurately represented within the simulation. Various numerical techniques exist to model rotation, differing in complexity and computational cost. Common approaches include moving wall, Multiple Reference Frame (MRF), Sliding Mesh or Rigid Body motion (RBM), Chimera method, and Immersed Boundary Method (IBM). While Chimera and IBM are frequently implemented in software such as OpenFOAM, they are omitted. Furthermore, sliding mesh is also discussed due to its dependency on the time domain. This section provides a brief overview of methodologies used within the limits of this project.

2.7.1 Moving Wall

The moving wall boundary condition is a simplified, steady-state approach for modeling rotationally symmetric bodies. Instead of mesh rotation, a tangential velocity is assigned to the wall boundary, calculated as the vector product of angular velocity and the position vector:

$$\vec{v} = \vec{\omega} \times \vec{r}$$

While this significantly reduces computational cost, it is restricted to axisymmetric geometries. When applied to non-axisymmetric features, cooling vanes in this case, the method becomes physically invalid. Consequently, it cannot capture the radial 'pumping effect' essential for cooling ventilated discs [12].

2.7.2 Multiple Reference Frame

The Multiple Reference Frame (MRF) method, or moving reference frame approach, provides a steady-state approximation of rotation without physical motion. A specific fluid zone is defined in a reference frame rotating at angular velocity $\vec{\omega}$ relative to the stationary inertial frame. Centrifugal and Coriolis accelerations are integrated into the momentum equations as source terms, allowing the simulation to capture the radial flow driven by rotation [13].

Although computationally efficient, MRF assumes a fixed relative position between rotating and stationary components. Its accuracy is highest when the flow is aligned with the axis of rotation. However, as Hobeika et al. cite [12], in cases with significant flow perpendicular to the rotation axis, velocity magnitude errors can exceed 40%. This occurs because MRF cannot resolve the transient interactions and pressure gradients (∇p) generated as rotating geometries rotate past stationary surroundings.

2.8 Heat Transfer

Thermal energy transfer occurs from regions of higher temperature to lower temperature through three distinct mechanisms: conduction, convection, and radiation. Conduction occurs when heat is transferred between two solid bodies in contact or through a solid medium. This process is governed by Fourier's Law of Heat Conduction, expressed as:

$$\dot{Q}_{cond} = kA \frac{T_1 - T_2}{\Delta x} \quad (2.10)$$

where T_1 and T_2 are the temperatures at the respective boundaries, and Δx is the thickness of the material. Radiation involves the transfer of heat through electromagnetic waves, requiring no physical medium or contact between the bodies. The maximum rate of heat emission is defined by the Stefan-Boltzmann Law:

$$\dot{Q}_{emit,max} = \sigma A_s T_s^4 \quad (2.11)$$

where σ is the Stefan-Boltzmann constant, and T_s is the absolute temperature of the surface. The third mechanism, and the primary focus of this research, is convection. Convection describes the heat transfer between a solid surface and a moving fluid, governed by Newton's law of cooling:

$$\dot{Q}_{conv} = hA_s(T_s - T_{ref}) \quad (2.12)$$

The Heat Transfer Coefficient (h) defines the rate of thermal energy transport from a surface to a fluid. Unlike material properties, h is a function of the flow field, fluid properties (density, viscosity), and geometry. While temperature can indirectly influence h by altering local fluid properties, it is not the primary driving factor. The influence of temperature on the heat transfer coefficient cannot be generalized. Equation (13) can be rewritten in a way to compute the heat transfer coefficient for a given heat flux and temperature difference:

$$h = \frac{\dot{Q}_{conv}}{A_s(T_s - T_{ref})} \quad (2.13)$$

Convection is categorized into two primary types: forced and natural (free). Forced convection occurs when fluid flow is induced by external means, such as the vehicle's motion, disc rotation in air, or the air flow in the test-bench, and its heat transfer rate is primarily driven by fluid velocity and turbulence. In contrast, natural convection is driven by buoyancy forces arising from density gradients near the heated body.

While disc brakes dissipate heat through conduction, radiation, and convection, forced convection is the dominant cooling mechanism during vehicle motion and

the exclusive focus of this research. Natural convection only becomes significant at a standstill when buoyancy-driven air movement takes over. Conduction into the hub occurs continuously but remains relatively minor compared to convection, while radiation only becomes a significant factor at elevated operating temperatures.[14]

2.9 CFD of a Disc Brake

Understanding the fluid dynamics within a ventilated disc is essential for establishing the pre-processing parameters and mesh requirements in the subsequent sections. The internal geometry of the ventilation cavity, as illustrated in Figure 2.7, dictates the primary cooling behavior of the rotor.

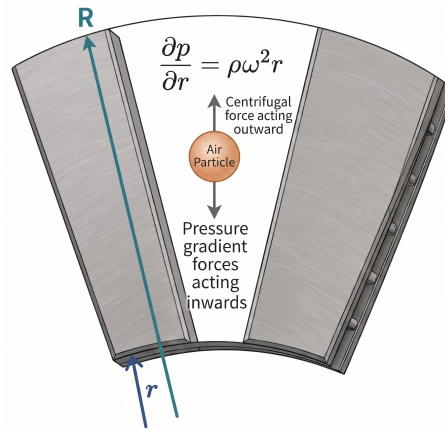


Figure 2.7: Schematic explanation of centrifugal action inside the ventilation cavity of a brake disc (The image was made using generative AI from a hand sketch)

A ventilated brake disc essentially functions as a centrifugal pump, drawing air from the hub region and expelling it radially toward the outer perimeter. This design significantly enhances the heat transfer by increasing the total surface area exposed to the air compared to a solid disc and also actively inducing mass flow through the rotor. As the disc rotates, the fluid trapped within the internal vanes is accelerated and flung outward by centrifugal forces. This results in a high-velocity discharge at the disc exit. A pressure differential creates a continuous suction effect, drawing fresh, ambient air into the cooling channels and maintaining a steady flow of coolant through the internal structure of the disc. [15]

As the fluid travels through the vanes, it absorbs significant thermal energy from the brake disc. This dissipation in thermal energy can alter the properties of the fluid and, in turn, the flow characteristics. This can be an important characteristic to be studied to understand the heat transfer mechanism and the influence of temperature on the flow characteristics inside the vanes.

3

Methodology

To ensure all the core objectives and goals of the research study were met, the project followed a structured methodology. This approach was chosen to deconstruct the objectives into sensible and actionable simple phases, establishing a progressive workflow in a way that the output of each phase serves either as a direct input or forms a fundamental basis to the next, illustrated in Figure 3.1

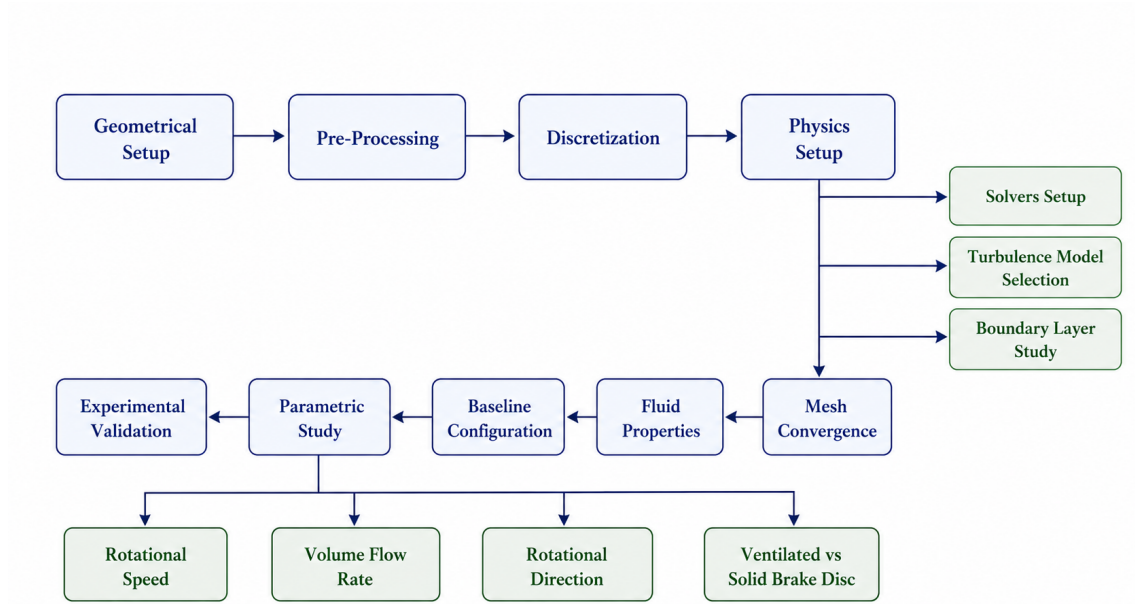


Figure 3.1: Workflow followed for the study

The workflow initiates from pre-processing, which encompasses geometric preparation, discretization parameters, and the definition of regions alongside their respective physical continua. To achieve suitable simulation accuracy while optimizing the utilization of high-performance computing resources, a mesh convergence study was carried out. Additionally, several preliminary simulations required to validate boundary conditions were performed during the pre-processing stage, which are documented and discussed within this section.

Once the computational domain was pre-processed and an appropriate mesh model was selected, simulations were carried out for various brake disc designs and various operational conditions. Initially, simulations were conducted to evaluate whether the common simplification of assuming constant fluid properties was physically justified

for this high-temperature study. Following this, the investigations of the distinct influences of rotational speed, inlet volumetric flow rate, and rotational direction on the localized convective heat transfer were analyzed. Furthermore, a comparative analysis between a ventilated brake disc and a solid brake disc configuration was performed to isolate and quantify the exact thermal dissipation contribution of the internal ventilation cavity.

To validate the numerical framework, the computational HTC's obtained from the fluid simulation stage, alongside real temperature and boundary data from the dynamometer test-bench, were imported into the finite element method (FEM) software Abaqus. A transient thermal FEM simulation was executed to compute the temperature distribution across the disc geometry under the identical testing profiles. Finally, the temperature fields from the physical test-bench, the CFD domain, and the structural FEM simulation were correlated to evaluate the accuracy, consistency, and alignment of the integrated numerical-experimental workflow.

4

Pre-Processing

4.1 Geometrical Setup

To achieve a correlation between the computational results and experimental data, it is important to replicate the entire test-bench environment rather than the standalone brake disc. While the primary focus lies on the brake disc and its geometric details, the surrounding test-bench is replicated in a simplified manner to capture the aerodynamic influence of each component, as shown in Figure 4.1. The simplifications are detailed as follows:

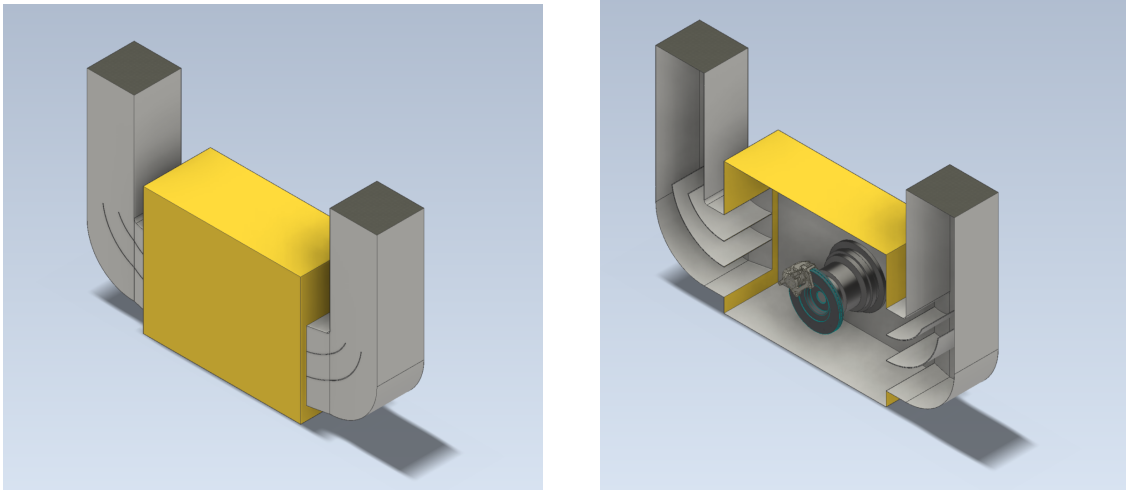


Figure 4.1: CAD models of the rig. Left: CAD replica of the test-bench, right: cross-section to visualize inside the enclosure

- The mechanical chuck used to rotate the disc is simplified into a series of coaxial cylinders. Their diameters closely match the actual dimensions to ensure the aerodynamic obstruction near the hub is correctly represented. (Figure 4.2)
- Minor dimensions and intricate features, such as small chamfers or fillets that do not significantly influence the global flow field, have not been modeled.
- Only the outer casing of the brake caliper (along with friction pads) is included in the simulation, as illustrated in Figure 4.2. Internal parts like the piston, etc., are not necessary. Because the caliper assembly partially encloses the brake disc and the vanes where the air exits, it constricts the local flow. The potential impact of this shielding effect will be discussed in the subsequent

sections.

- As the study focuses on the convective cooling of the rotor, the walls and the parts of the test-bench (for instance, the caliper) do not have material properties assigned to them. Instead, they are set to the ambient room temperature of 20 °C.

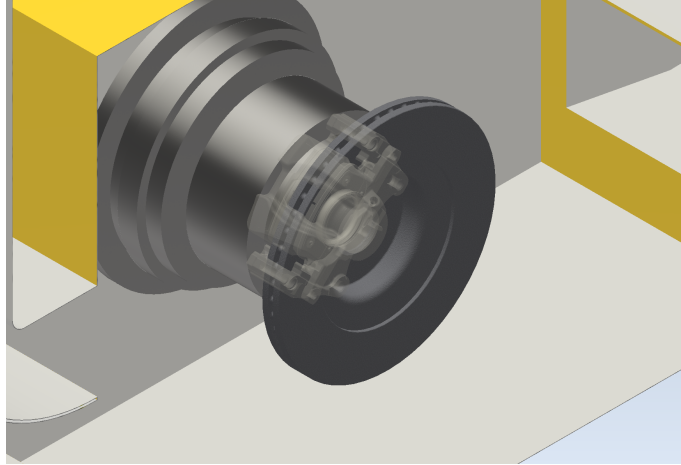


Figure 4.2: 3D Model of the chuck inside the enclosure

The CAD models of the brake discs and calipers were provided by VCPC as STEP files. A brief description of the geometrical dimensions of the brake discs used for this study is tabulated below (Table 4.1), and the internal structure of the vanes in the case of ventilated brake discs is visualized in Figure 4.3.

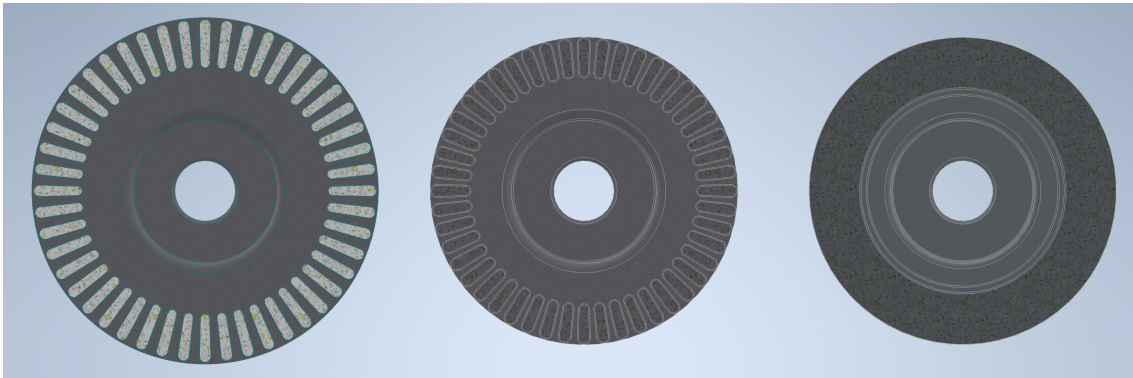


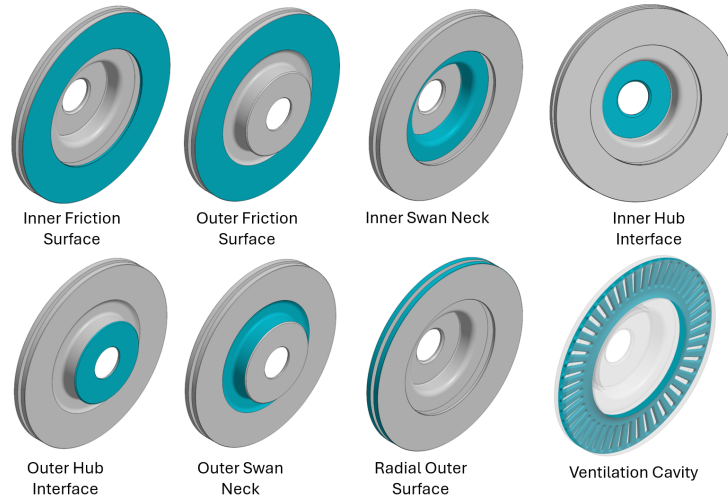
Figure 4.3: Brake discs used for this study. Left: ventilated brake disc 1, middle: ventilated brake disc 2, right: solid brake disc 3

Brake Disc 1 and Brake Disc 2 share the same internal vane geometry and thickness of the friction planes, but the external diameter and the thickness of the vanes are the primary variables. Conversely, Brake Disc 2 and Brake Disc 3 share the same diameter but differ in construction type (ventilated versus solid brake disc).

Table 4.1: Geometrical overview of the brake discs

S. No.	Brake Disc ID	Type of Brake Disc	Diameter (mm)	Thickness (mm)
1	Brake Disc 1	Ventilated Brake Disc	366	28
2	Brake Disc 2	Ventilated Brake Disc	325	28
3	Brake Disc 3	Solid Brake Disc	325	13.6

The brake discs and their respective caliper assemblies were first assembled in Autodesk Inventor and exported as STEP files. These files were then imported into the ANSA pre-processor to define the so-called PIDs (Property IDs). A PID, in this context, represents a collection of CAD surfaces or features grouped for computational purposes. Since it would be extremely difficult to calculate the HTC at each discrete point of the surface, a surface split-up was desired so that the HTC could be averaged over the surface. A simplified assumption is that the HTC is constant over a surface. The existing FEM model already has a surface split up, according to which the surfaces can be segregated in this project. In addition to the eight primary surfaces defined for HTC calculation, two additional surfaces, inner and outer interfaces at the inlet and outlet of the ventilation cavity, respectively, were generated using the 'fill' or 'parametric extrude' function in ANSA to create interfaces. These interfaces, acting as transparent boundaries, serve the fluid flow between the air region and the ventilation cavity of the brake discs, in the case of ventilated brake discs. The surface split-up is shown in Figure 4.4.

**Figure 4.4:** Surface segmentation used for localized HTC mapping

In the case of a solid brake disc, the ventilation cavity, inner and outer interfaces are not present.

Once the PIDs are assigned, the brake discs have to be exported as follows:

- **Ventilated Disc:** Three parts - the rotating fluid region (ventilation cavity), the surface of the brakes and caliper assembly. The reason for the need to divide the ventilated disc into surface and vane domains will be motivated in the upcoming subsection.
- **Solid Disc:** Two parts - brake surfaces and caliper assembly

Upon importing the parts and the test rig into StarCCM+, it has to be made sure that the axes of the brake disc, caliper, and the fixture are coaxial with the chuck. A CAD Boolean operation is performed where the caliper assembly and the surfaces of the brake disc are subtracted from the rig. This creates a 'negative mold' of the parts where the air can flow, thus resulting in a region for the fluid to flow.

4.2 Discretization

Following the geometric preparation, the domain must be discretized into a computational mesh. Since the CAD surfaces were pre-meshed and validated for topological errors in ANSA, no further surface preparation was required within Simcenter StarCCM+. For the volume mesh, the Polyhedral Mesher was selected over the trimmed cell mesher. As cited by Wang et al. [16], polyhedral cells generally offer superior performance compared to tetrahedral meshes (visualized in Figure 4.5), as they possess more neighbors, allowing for more accurate gradient calculations and better convergence with fewer total cells. To ensure a high-quality starting point for the volume discretization, the Surface Remesher and Automatic Surface Repair models were also enabled.

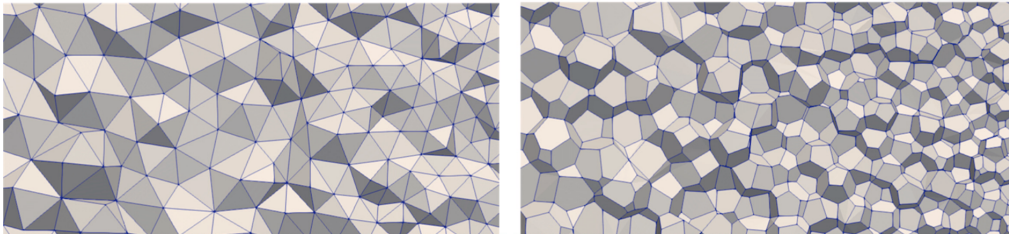


Figure 4.5: Types of cells used for CFD. Left: tetrahedral cells, right: polyhedral cells [16]

The polyhedral mesh is applied throughout the entire fluid domain; however, the resolution is locally refined to capture the relevant physics. Given that the thermal boundary layer is the primary driver of heat transfer, a high-density mesh is required near the rotor. To achieve this, a surface offset operation was performed on all disc surfaces at a distance of 75 mm , creating a conformal enclosure that follows the disc's geometry. Furthermore, since a brake disc against the air flow generates a significant wake, a secondary refinement zone was implemented downstream. This wake region maintains a relatively fine mesh, though slightly coarser than the immediate 75 mm offset, to capture the high gradients in velocity and temperature typical of turbulent separation. Outside of these critical zones, a coarser mesh is utilized to reduce the

overall computational cost. The mesh parameters will be discussed in the following subsection.

StarCCM+ offers diverse settings to vary the mesh structure. The base size refers to a value that serves as a relative value for other mesh parameters. The target surface size defines the desired size of the polyhedral cell, and the minimum surface size defines the minimum size of the cell on the surface of a solid body. The prism layers and their configurations will be explained in the upcoming subsection. The maximum tet size defines the maximum size of the polyhedral cells in a computational domain. The surface growth defines how much the cells should develop from the surface cell size. Analogous to surface growth rate, the volume growth rate defines how much the polyhedral growth should be in the computational domain.

4.3 Physical and Boundary Models

The physics for the simulation, along with the boundary and initial conditions, is discussed in this subsection. The important point is that the brake disc was not treated as a solid region, but rather as surfaces, since convective cooling does not necessarily need the material properties. Furthermore, it is computationally efficient to consider the brake disc as a surface and simulate only the fluid region. The physics solver models used for this work are listed as follows:

Table 4.2: Physics solvers and models enabled

Category	Selected Model
Matter and properties	Gas
	Ideal gas
Flow Solvers	Coupled Flow
	Coupled Energy
	Gradients
Spatial Domain	Three-dimensional
Time domain	Steady
Viscous regime	Turbulent
Turbulence Models	Reynolds-Averaged-Navier-Stokes
Wall Treatment	Wall Distance
	All $y+$ Wall Treatment

As mentioned in the subsection 4.1, the segregation into the vane domain and surface domain is due to the fact that the MRF rotation modeling is employed. In that case, the vanes are made to rotate in a different reference frame as opposed to the

fluid region reference frame. Two regions are created, one as a result of the Boolean subtraction operation carried out and the other as the rotational domain. As explained in the section 4.1, the interface surfaces have to be assigned the interface boundary condition. The surfaces' inner and outer interfaces have been linked together to create flux boundaries between the vanes and the rest of the fluid domain of the test-bench (Interface 3 and Interface 4 in Figure 4.6). The various boundary conditions and initial conditions for the two domains are defined as follows:

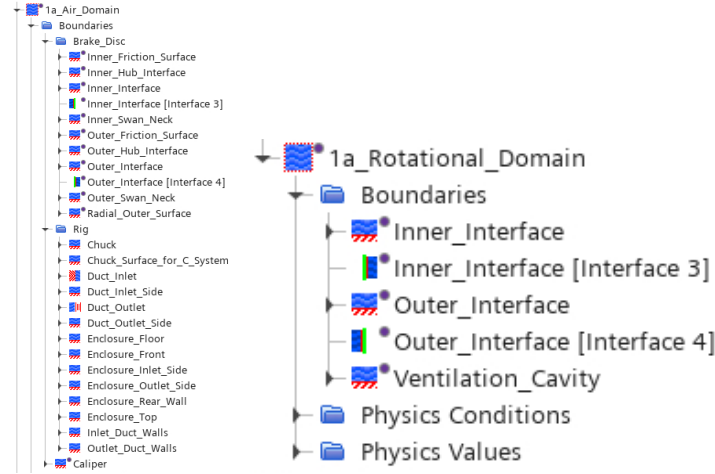


Figure 4.6: Defined regions from the surfaces. Left: Fluid domain, right: rotational domain

A steady-state solver was selected because no transient cooling is simulated, matching the requirements of the MRF rotational model. An ideal gas solver was implemented to capture the local fluid density variations induced by the heated surfaces. Since the HTC is driven primarily by local temperature and flow dynamics rather than material properties, this approach ensures that air density remains a variable.

4.3.1 Choice of Turbulence Model and Boundary Layer Study

Selecting an appropriate turbulence model requires balancing physical fidelity with computational cost. This choice directly influences the boundary layer resolution, mesh prism layer configuration, and wall treatment, which can otherwise under- or over-predict convective heat transfer results.

The two primary options for this study are the two-layer realizable $k-\epsilon$ and the $k-\omega$ SST models. While previous brake cooling studies [13, 17] utilized the realizable $k-\epsilon$ model, they lacked experimental validation to justify the selection. Conversely, a comprehensive comparative study by Reddy et al. [18] demonstrated that the $k-\omega$ SST model predicts the flow field with superior accuracy. To evaluate boundary layer behavior and finalize this selection, baseline configurations were established using the parameters detailed in Table 4.3. Because the $k-\omega$ SST model effectively combines the advantages of $k-\epsilon$ with the advantages of $k-\omega$, it was selected for use throughout this study.

Table 4.3: Prism Layer base Parameters

Mesh Parameter	Air Domain	Rotational Domain
Number of Prism Layers	20	10
Prism Layer Total Thickness (mm)	4	2
Prism Layer Stretching	1.2	1.2

Once the turbulence model was selected, its expected y^+ validity was verified, as shown in Figure 4.7. The values remain predominantly within the target range of 1, with only a marginal percentage extending up to 5. To confirm that this prism layer configuration accurately resolves the thermal boundary layer, a temperature scalar scene with the superimposed grid layout was generated across the outer surfaces and internal vanes.

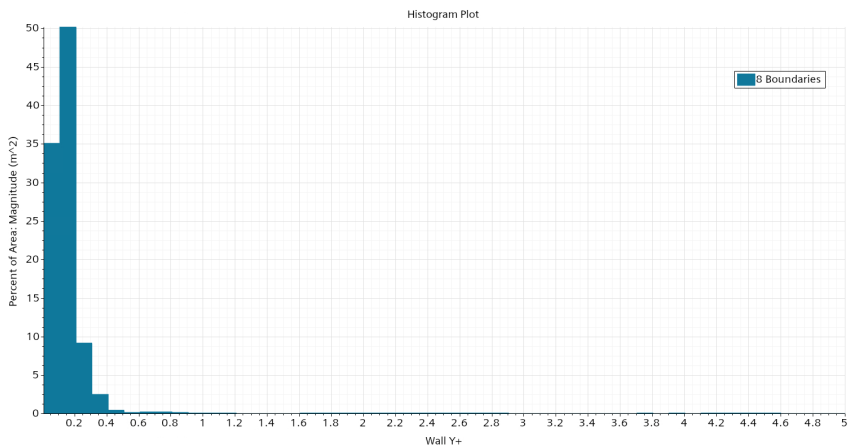


Figure 4.7: Wall y^+ plot against area of the brake surfaces

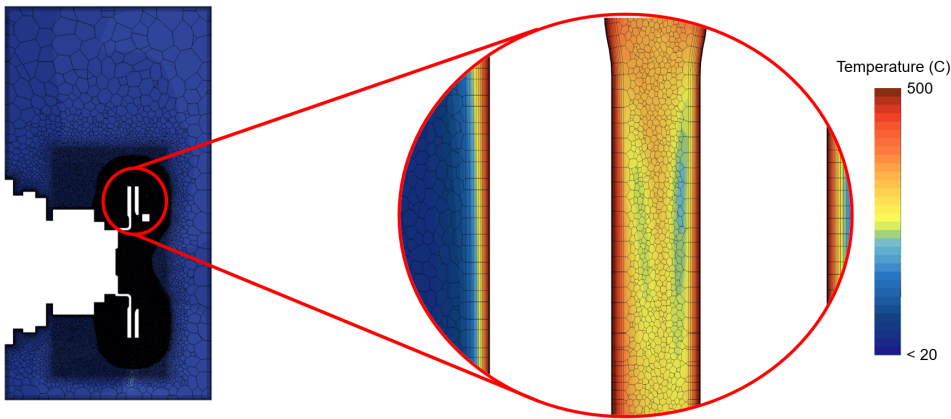


Figure 4.8: Computational mesh resolution within the fluid domain. Left: cross-sectional front view, right: magnified view of the prism layer grid resolving the near-wall thermal boundary layer inside the cooling vanes

Figure 4.8 shows that the near-wall prism layers successfully resolve the strong gradients within the thermal boundary layer. Conversely, the initial 20-layer, 2 mm thickness proved excessive for the thin boundary layer generated by the accelerated flow inside the internal vanes. To balance computational efficiency while maintaining the recommended minimum of 10 cells between vane walls to accurately capture local gradients, both the prism layer count and total thickness were reduced. The finalized meshing parameters are detailed in Table 4.4.

Table 4.4: Mesh Parameters

Mesh Parameter	Fluid Domain	Rotational Domain
Base Size (mm)	17.5	4
Target Surface Size (mm)	1.75	1
Minimum Surface Size (mm)	0.875	0.6
Surface Growth Rate	1.3	1.3
No of Prism Layers	18	14
Prism Layer Total Thickness (mm)	2	1
Prism Layer Growth rate	1.2	1.2
Maximum Tet Size (mm)	0.875	0.4

4.3.2 Motion Specification

Air Domain

The air domain motion is specified as stationary, utilizing the default laboratory reference frame.

Rotational Domain

A rotating reference frame is established for the rotation domain, with its origin and coordinate system aligned with the center of the mounting chuck. Its angular velocity is controlled via the user-defined parameter '*rotational_speed*'. Following the Multiple Reference Frame (MRF) approach, the region is set to be stationary relative to this rotating reference frame.

4.3.3 Thermal Boundary Conditions

Air Domain

The temperature for all stationary test-bench surfaces and auxiliary components, such as the brake caliper, is set to a constant 20°C. The brake disc surfaces are assigned a variable parameter, '*brake_disc_temperature*', to facilitate thermal sensitivity analysis.

Rotational Domain

The internal ventilation cavity surfaces are assigned the same temperature parameters as the external brake disc surfaces. The fluid interfaces between the regions are

governed by an adiabatic boundary condition. Because these interfaces are physically transparent and serve solely as flux boundaries between domains, any non-interfaced cells (under 0.001%) are designated as adiabatic to prevent nonphysical local temperature effects.

4.3.4 Domain Inlet and Outlet Conditions

The test-bench inlet is assigned a 'mass flow inlet' boundary condition, where the mass flow rate is dynamically calculated from the variable '*volume_flow_rate*' and the local inlet fluid density. The outlet is defined as a 'pressure outlet' maintained at ambient pressure. The above-mentioned boundary conditions are summarized in the table 4.5.

Table 4.5: Boundary and initial conditions used in the simulation

S. No.	Parts / Surfaces	Boundary / Initial Condition	Value / Property	Motion Specification
1	Duct Inlet	Mass flow inlet	variable defined using formula	Fixed
2	Duct Outlet	Pressure outlet	1 atm	Fixed
3	All the test-bench surfaces	Static Temperature	20 °C	Fixed
4	Chuck surfaces	Static ambient temperature	20 °C	Tangential Velocity: Local Rotation rate
5	Brake surfaces except ventilation cavity and interface regions	Static temperature	variable defined as parameter	Tangential Velocity: Local Rotation rate
6	Ventilation cavity	Static temperature	variable defined as parameter	Rotating Reference Frame

4.4 Mesh Convergence Study

The objective of the mesh convergence study is to establish an optimal balance between computational cost and the accuracy of the physical parameters. It is necessary to identify the point at which further mesh refinement yields negligible changes in the results, ensuring that the solution is independent of the grid resolution.

For this study, four volumetric refinement zones were utilized: three within the stationary air domain, consisting of the near-wall offset zone, the wake zone, and the far-field domain, and one encompassing the ventilation cavity within the rotational domain. A series of simulations was conducted by systematically varying the poly-

hedral base size across both the fluid and rotational domains. The parameters for the different mesh configurations are summarized in Table 4.6.

Table 4.6: Mesh convergence cases

Case No	Fluid Domain (cell size in mm)			Rotational Domain (cell size in mm)		No of cells (in millions)
	Base Size	Offset zone (5%)	Wake Zone (10%)	Base size	Tetrahedral Size (10%)	
1	10	0.5	1	1	0.1	647.68
2	15	0.75	1.5	2	0.2	371.98
3	17.5	0.875	1.75	2	0.2	104.78
4	20	1	2	2	0.2	82.37
5	22.5	1.125	2.25	4	0.4	38.78
6	25	1.25	2.5	4	0.4	31.02
7	40	2	4	4	0.4	12.34
8	50	2.5	5	4	0.4	9.24
9	50	2.5	5	6	0.6	6.84
10	75	3.75	7.5	8	0.8	2.86
11	100	5	10	12	1.2	1.41

To evaluate convergence, several key parameters were monitored, including the drag coefficient (C_d), mass flow rate, and surface heat flux. In this study, the drag coefficient does not refer to the vehicle's aerodynamic drag, but rather to the aerodynamic torque opposing the rotation of the brake disc. These values were plotted against the total cell count to visualize the mesh-independent solution. The resulting plots are presented in Figures 4.9 and 4.10.

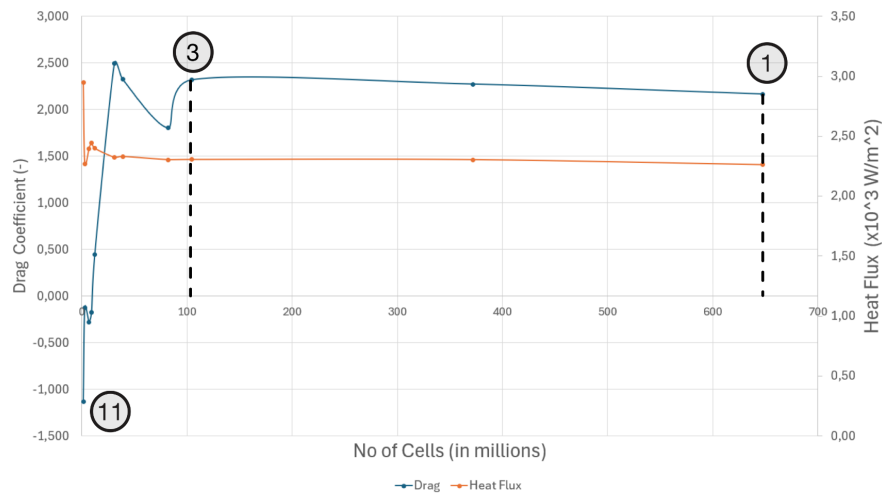


Figure 4.9: Heat flux of ventilation cavity and drag coefficient against the number of cells

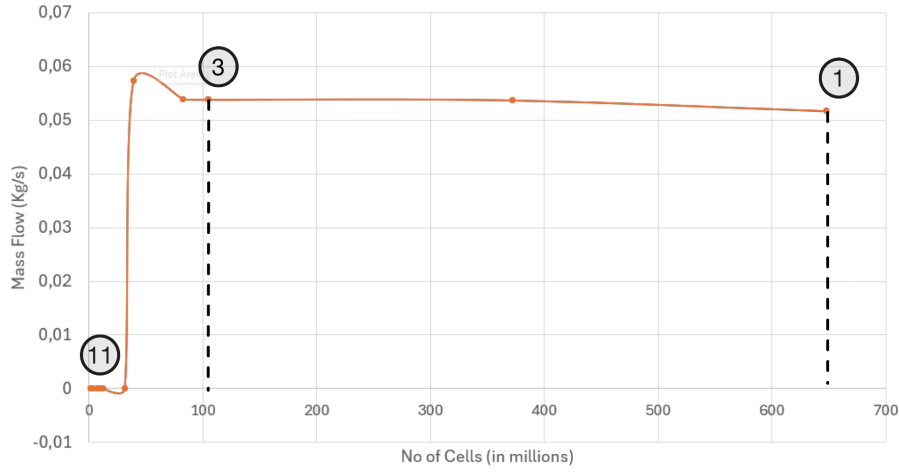


Figure 4.10: Averaged mass flow rate in the vanes against the number of cells

The plots reveal that the monitored parameters are independent of the mesh size from Case 3 onward. While the most refined configuration (Case 1, 600 million cells) provides the highest resolution, its extreme computational demand exceeds the necessary scope of this research. In contrast, the coarsest configurations (Cases 9–11) fail to resolve the intricate geometry of the ventilation cavity, leading to nonphysical discretization errors such as negative drag values.

Case 3, with 104 million cells, was selected as the optimal production mesh. It provides a robust compromise between numerical accuracy and computational efficiency, ensuring the flow physics within the internal vanes is fully captured without unnecessary overhead. The final mesh configuration for the computational domain is visualized in Figure 4.11

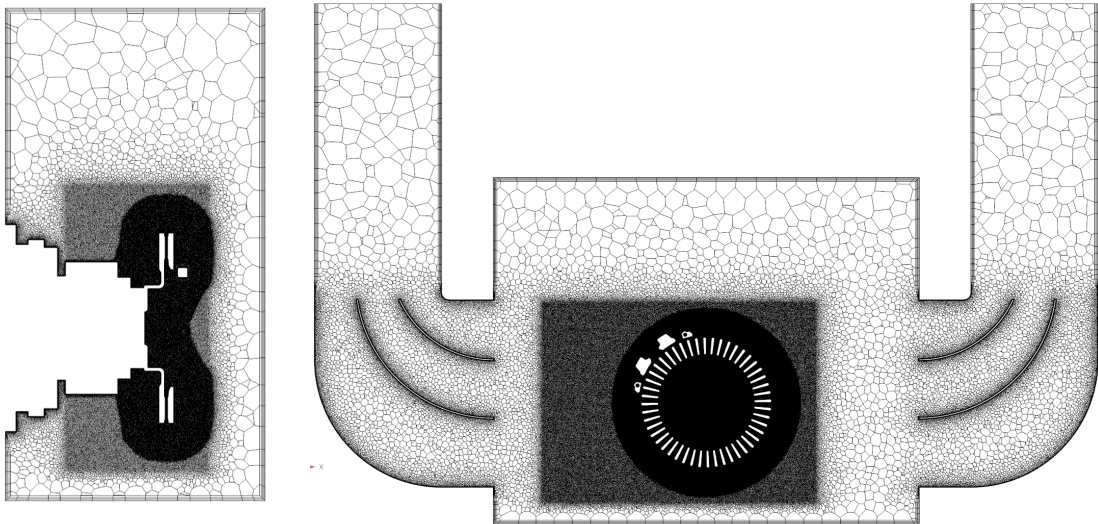


Figure 4.11: Cross sections of meshed computational domain. Left: front view, right: side view

5

CFD Simulation Cases

This section examines the parameters influencing the HTC of the brake disc, replicating laboratory dynamometer conditions. To correlate simulation inputs with physical vehicle velocities, rotational speeds of the brake disc were calibrated using a dynamic rolling radius. Parametric evaluations were conducted across a wide range of rotational speeds and brake disc temperatures, focusing primarily on two standard test-bench inlet airflow rates: 300 m³/hr and 1200 m³/hr (representing the volumetric amount of air introduced as an input to the test bench). Additionally, a separate case study isolates the direct effect of an increasing volumetric inlet airflow rate to the test-bench on heat dissipation, providing the benefit of extending the baseline dataset or accommodating alternative volumetric values for future testing procedures.

Thermal evaluations began at a baseline of 30°C, replicating the experimental test procedure’s starting temperature, and then incrementally increased from 100°C to 800°C in 100°C increments. While 800°C represents an extreme scenario rarely encountered in passenger vehicles, its inclusion ensures model robustness and accurate interpolation across the full thermal spectrum. All subsequent parametric studies utilize this temperature range unless specified otherwise.

5.1 Case Setup: Temperature-Dependent Fluid Properties

A critical prerequisite to establishing the baseline configuration for this parametric study is assessing the extent to which the intrinsic properties of air, specifically thermal conductivity (k), dynamic viscosity (μ), and specific heat capacity (C_p), influence the resulting heat transfer characteristics. This sensitivity analysis is vital to understand the influence, due to the rigorous brake testing at the dyno and a significant increase in temperatures.

If the deviation between constant and variable property models is negligible, the former may be utilized to reduce computational overhead. However, existing research indicates that neglecting temperature-dependent variations can result in significant predictive errors, particularly at high thermal gradients [19, 20]. To quantify this impact, the study evaluates whether the increased kinematic viscosity and enhanced thermal conductivity at elevated temperatures offset or amplify the cooling rate.

Convective heat transfer is primarily driven by fluid properties evaluated at the film temperature (T_f):

$$T_f = \frac{T_s + T_\infty}{2} \quad (5.1)$$

As established in subsection 2.6, the HTC depends on the Reynolds and Prandtl numbers, which are functions of these temperature-dependent properties. To account for these variations, Sutherland’s Law was implemented for thermal conductivity and dynamic viscosity, while a polynomial fit was utilized for specific heat, leveraging the physical models within StarCCM+.

Two comparative cases were simulated: one with constant properties of air evaluated at 20°C and a second utilizing these empirical laws to capture the temperature-dependent physics.

Table 5.1: Case setup and boundary conditions for the fluid properties

Property	Case 1	Case 2
Dynamic Viscosity	$1.855 \times 10^{-5} Pa/s$	Sutherland’s Law
Thermal Conductivity	0.0260305 W/m-K	Sutherland’s Law
Specific heat	1003.62 J/kg-K	Polynomial Fit Curve in T

5.2 Case Setup: Baseline Configuration

Following the selection of suitable physical models for fluid properties, a baseline configuration was established to characterize the fundamental thermal behavior of the brake disc. This baseline serves a dual purpose. First, it provides a benchmark for evaluating the fundamental convective performance and heat flux distribution, and secondly, it acts as a reference point to ensure that subsequent parametric variations, such as changes in geometry or rotational direction, yield logically consistent trends.

For this control case, a vehicle speed of 180 km/h (equating to an rotational speed of 1281.78 rpm) and a volumetric inlet airflow rate of 300 m³/hr were selected. To validate the simulation’s accuracy, several critical field functions were monitored:

- **Internal Flow Dynamics:** The velocity profiles and mass flow rates to and out of the ventilation cavity were analyzed to confirm the presence of the expected centrifugal pumping effect.
- **Thermal Distribution:** Area-averaged surface heat flux and heat transfer were mapped across the disc’s functional surfaces to identify primary cooling zones.
- **Convective Performance:** The area-averaged HTCs were evaluated to ensure the boundary layer physics were being captured correctly at the interface between the rotating disc and the forced airflow.

5.3 Case Setup: Variable Rotational Speeds

The rotational speed of the brake disc at the test bench correlates directly to vehicle velocity. This parametric study investigates the heat transfer coefficient across various speeds, ranging from 30 km/h to 280 km/h in increments of 50 km/h. Since StarCCM+ requires the rotational speed in rpm as input, the following formula was applied:

$$N_{brake} = \frac{\omega \cdot 60}{2\pi} \Rightarrow \frac{v_{veh} \cdot 1000}{3600} \cdot r_{dyn} \cdot \frac{60}{2\pi}$$

While a vehicle speed of 280 km/h represents an upper limit for most passenger vehicle operations, it was simulated to provide additional data points and enhance the stability of the interpolation curves for specific target speeds. Six cases were configured for investigation at volumetric inlet flow rates to the test-bench of 300 m³/h and 1200 m³/h, respectively, as summarized in the table below:

Table 5.2: Case setup for influence of rotational speed on heat transfer

Case No.	Vehicle Speed (km/h)	Rotational Speed (rpm)
1	30	213.63
2	80	569.68
3	130	925.73
4	180	1281.78
5	230	1637.83
6	280	1993.88

5.4 Case Setup: Variable Rotational Direction

Although the driving unit of the dyno test bench cannot rotate in the counterclockwise direction and the vanes of the ventilated brake disc are axisymmetric, lacking predefined orientation, the complete test-bench assembly is asymmetric. This study was intended to determine whether the asymmetric positioning of the caliper relative to the disc, as well as the presence of the chuck and its impact on the airflow, influences the heat transfer characteristics. For this evaluation, the same speeds as in 5.3 are considered but also in the counterclockwise direction. Nevertheless, only the rig inlet airflow rate of 300 m³/hr was considered.

5.5 Case Setup: Variable Inlet Airflow Rate

It was established previously that the fluid properties act as the main driving factor for heat transfer. Hence, it is important to understand how the brake disc performs

in terms of heat transfer characteristics when it is exposed to various airflow rates. The inlet airflow rates are varied over a wide range from 150 m³/hr to 1200 m³/hr. In addition to that, the speed of the vehicle is fixed at 180 km/h (1281.78 rpm). The case setup is shown in Table 5.3. Analogous to rotational speed, StarCCM+ takes the input for the inlet airflow rate in terms of mass flow rate. Since the volumetric airflow rate to the test-bench was provided according to the test procedure, a formula was defined that converts the volumetric airflow rate to mass flow rate for the test bench. The formula is expressed as:

$$\dot{m} = \frac{\dot{V} \cdot \rho}{3600}$$

Table 5.3: Case setup for influence of volume flow rate speed on heat transfer

Case No.	Inlet Airflow Rate (m ³ /hr)	Inlet Mass flow rate (kg/s)
1	150	0.05
2	300	0.1
3	500	0.16
4	700	0.23
5	900	0.30
6	1200	0.40

5.6 Case Setup: Brake Design

This section evaluates the cooling efficiency of the ventilation cavity by comparing brake disc 2 (ventilated) and brake disc 3 (solid), as illustrated in Figure 5.1. A primary objective of this subsection is to determine how the cooling load previously handled by the ventilation cavity is redistributed among the remaining external surfaces when the internal vanes are removed, and to quantify the extent of this redistribution. Given that the underlying influences of rotational speed and ambient airflow were established in previous sections, this comparison is conducted at a single control point corresponding to 180 km/h vehicle speed and 300 m³/hr inlet volumetric flow rate, across the specified temperature range to isolate the explicit impact of internal geometry on thermal performance.

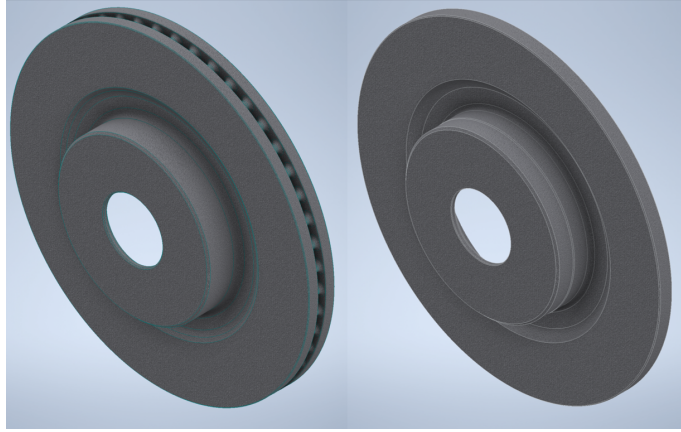


Figure 5.1: Brake disc 2 and brake disc 3

The comparative analysis focuses exclusively on the HTC's and absolute heat transfer. Aerodynamic metrics such as mass flow rate and velocity from the vanes are omitted, as they are non-applicable to the solid disc geometry. Furthermore, this evaluation cannot be perfectly parameterized because the overall disc thickness is reduced and a different brake caliper assembly is utilized for each configuration. Therefore, two distinct simulation strategies are carried out to isolate the exact thermal contribution of the ventilation cavity. In the first approach, the interface surfaces at the inlet and outlet of the brake vanes (explained in the section 4.1) are treated as adiabatic walls, preventing them from acting as flux boundaries for the fluid. Under these boundary conditions, the ventilated disc serves as a solid brake disc, permitting a direct comparison across shared external surface zones.

In the second approach, the physical reality of the solid disc is modeled directly, where the absence of brake's internal vanes is supplemented by a reduction in profile thickness. Consequently, while the solid disc possesses a lower thermal mass than its ventilated counterpart, it is also expected to exhibit reduced energy dissipation due to the absence of a fluid-exposed area. This second case involves a direct comparison of brake disc 2 with brake disc 3, evaluating the combined thermal impact of the different caliper geometries and the reduced external surface area. The configurations and boundary conditions for this comparative study are summarized in Table 5.4.

Table 5.4: Case setup for influence of ventilation cavity on heat transfer

Case No.	Comparison of	Compared with
1	Brake disc 2 (Ventilated)	Brake disc 2 (adiabatic interface surfaces)
2	Brake disc 2 (Ventilated)	Brake disc 3 (Solid brake disc)

6

Results and Discussion

Simulations were run until a converged solution was achieved, with final results averaged over the last 300 iterations to eliminate residual oscillation error. To comprehensively analyze the factors governing the temperature-dependent thermal dissipation of the brake disc, surface-averaged HTC_s were extracted according to the segmentation mentioned in the section 4.1. While primary emphasis remains on studying these surface-averaged HTC_s as a function of the brake surface temperature, absolute heat transfer and heat flux were also evaluated to understand the thermal behavior. However, to maintain the core scope of this study on localized HTC_s, the absolute heat transfer and heat flux plots are compiled within the Appendix (except for the baseline configuration and ventilation cavity study). Scalar scenes of the same were also generated to visualize the heat transfer phenomenon and to study the flow characteristics. It can be beneficial to study mass flow, volume flow, velocity, and pressure in the ventilation cavity to understand the cooling performance of the brake disc.

6.1 Evaluation of HTC at Variable Fluid Properties

The heat transfer scalars, namely heat transfer and HTC of the ventilation cavity, visualized in Figure 6.1, seem to be under-predicted under the assumption of constant fluid properties. This discrepancy arises because the heat transfer is intrinsically linked to the film temperature (equation 5.1).

By assuming constant properties, the solver universally applies the density and viscosity values of the ambient reference state (20°C) across the entire temperature range, failing to account for the expansion of the air film at elevated temperatures. The percentage deviation in heat transfer between the constant and variable property models, shown in Figure 6.2, scales non-linearly with temperature. It exhibits a steep initial growth during early temperature changes before the slope gradually tapers off at elevated temperatures. While the deviation is a negligible 0.7% at 30°C, it reaches a substantial maximum of 19.6% at 800°C. This heavy variance underscores the critical importance of utilizing variable fluid properties for thermal analysis in this domain, a necessity supported by established literature [19, 20]. As a result of this, the subsequent parametric study was conducted using variable fluid properties governed by the previously defined gas laws.

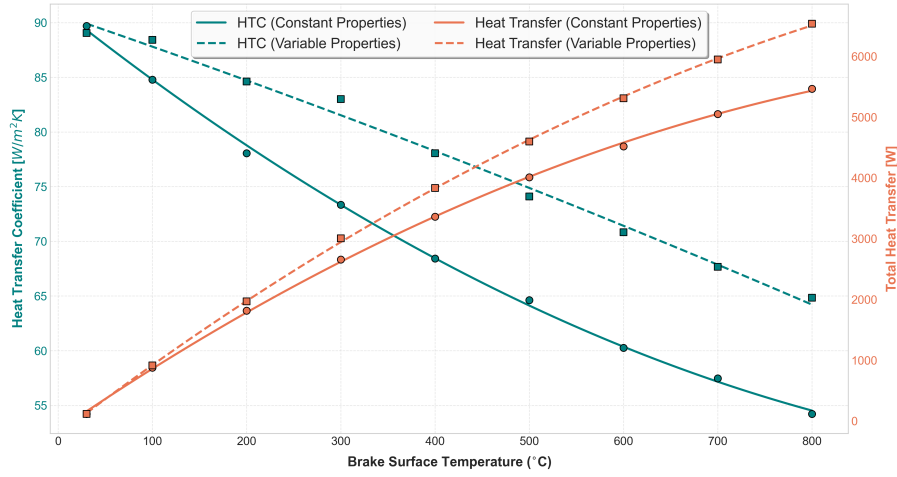


Figure 6.1: Influence of fluid properties on surface-averaged HTC and heat transfer of the ventilation cavity (180 km/h, 300 m³/h)

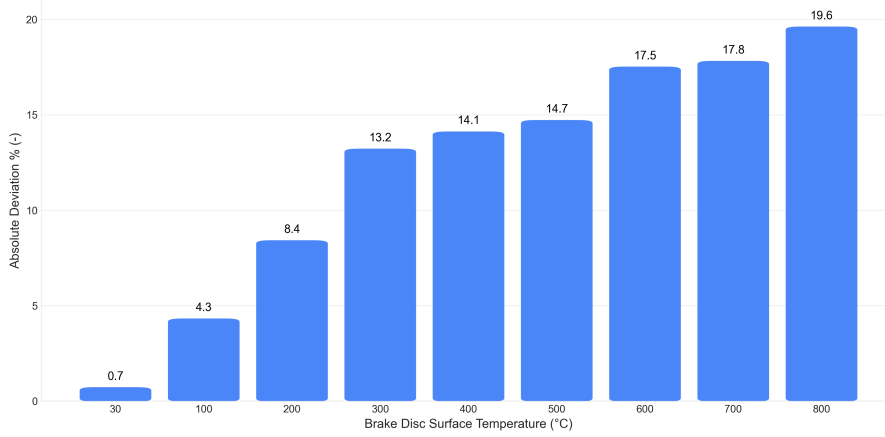


Figure 6.2: Percentage deviation of the HTC values with variable properties as the baseline

6.2 Evaluation of HTC of Baseline Configuration

Initial results, visualized in Figures 6.3, indicate that the HTC for all surfaces decreases as the surface temperature rises, except for the outer hub interface, which exhibits nearly constant. The ventilation cavity and the outer hub interface consistently maintain the highest HTC and heat flux, as shown in Figure 6.4. Furthermore, the inner friction surface exhibits the fourth highest heat flux, despite its constant exposure to ambient air and a larger surface area than the radial outer surface. Therefore, evaluating the total heat transfer rate (in Watts) alongside HTC and heat flux is necessary to accurately assess the cooling mechanism.

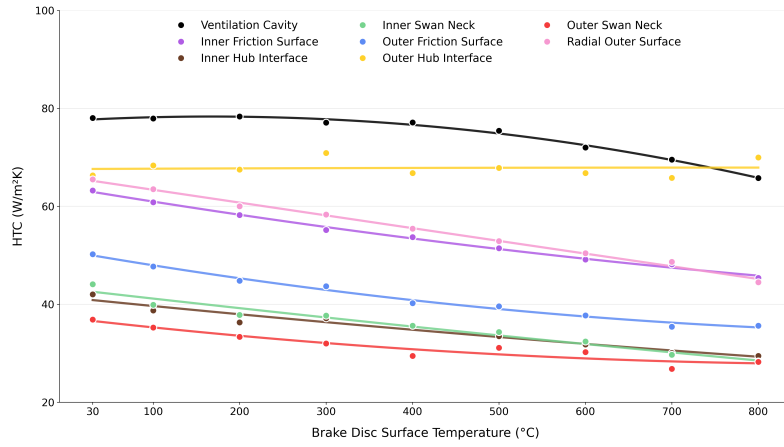


Figure 6.3: Surface averaged HTC as a function of surface temperature (180 km/h, 300 m³/h)

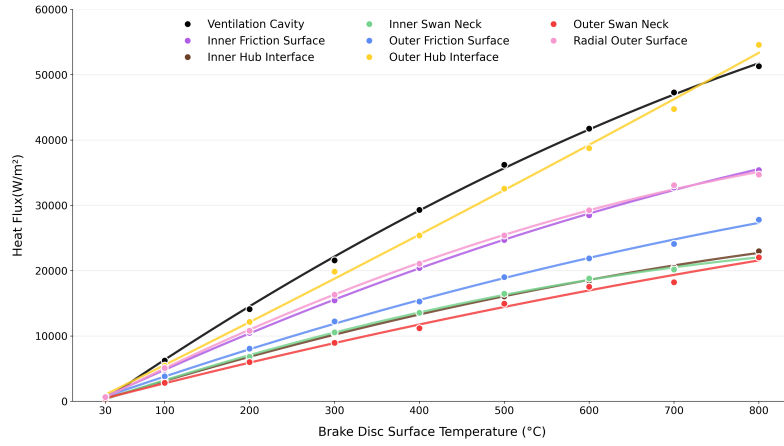


Figure 6.4: Surface averaged heat transfer as a function of surface temperature (180 km/h, 300 m³/h)

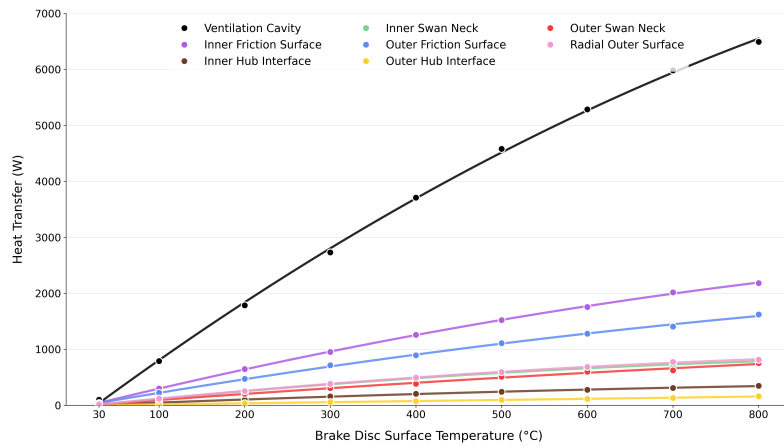


Figure 6.5: Surface averaged heat flux as a function of surface temperature (180 km/h, 300 m³/h)

The heat transfer curves in Figure 6.5 confirms that the ventilation cavity is the primary cooling driver, followed by the friction surfaces, radial surfaces, swan neck, and hub interfaces. This hierarchy emphasizes the effectiveness of the internal vanes, which utilize centrifugal pumping to maximize thermal energy removal.

The assumption that the HTC (h) is independent of temperature is a common simplification. Similar numerical investigations into brake disc simulations have reported an identical reduction in convective efficiency as the temperature difference increases [13]. Furthermore, Zhao et al. [21] developed a generalized expression for convective heat transfer that corroborates this downward trend, providing a theoretical foundation for the phenomenon observed in this study. To understand the physics governing this decrease, both theoretical analysis of dimensionless numbers and three-dimensional cross-sectional planes are utilized.

Theoretical explanation

The behavior of the HTC, as previously established, is fundamentally tied to the temperature-dependent nature of fluid properties. Analyzing these effects through dimensionless numbers clarifies the underlying mechanics.

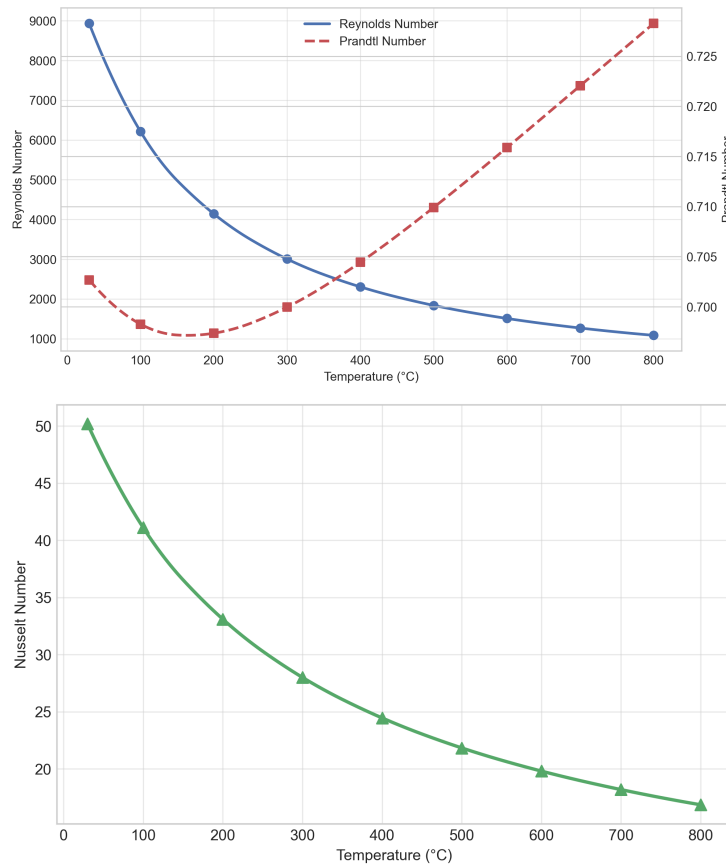


Figure 6.6: Dimensionless numbers against temperature. Left: Reynolds and Prandtl number, right: Nusselt number

The plot in Figure 6.6 demonstrates that for increasing temperature, the air's dynamic viscosity increases and (μ) and density (ρ) decreases significantly, thereby

causing the Reynolds number to decrease substantially. While the Prandtl number (Pr) technically increases with temperature, driven by shifts in dynamic viscosity, specific heat, and thermal conductivity, this change occurs primarily in the second decimal place.

The Nusselt number (Nu), which serves as the dimensionless representation of the heat transfer coefficient, is a function of both Re and Pr . When Nu is plotted against temperature, a steady decline is observed, as in Figure 6.6. This is due to the fact that the strong decrease in the Reynolds number (representing reduced inertial forces and increased viscous damping) outweighs the marginal effects of the Prandtl number and the increase in thermal conductivity. Consequently, the thermal degradation of the convective heat transfer coefficient (CHTC) can be directly linked back to this fundamental decrease in the Nusselt number.

Three-dimensional flow analysis

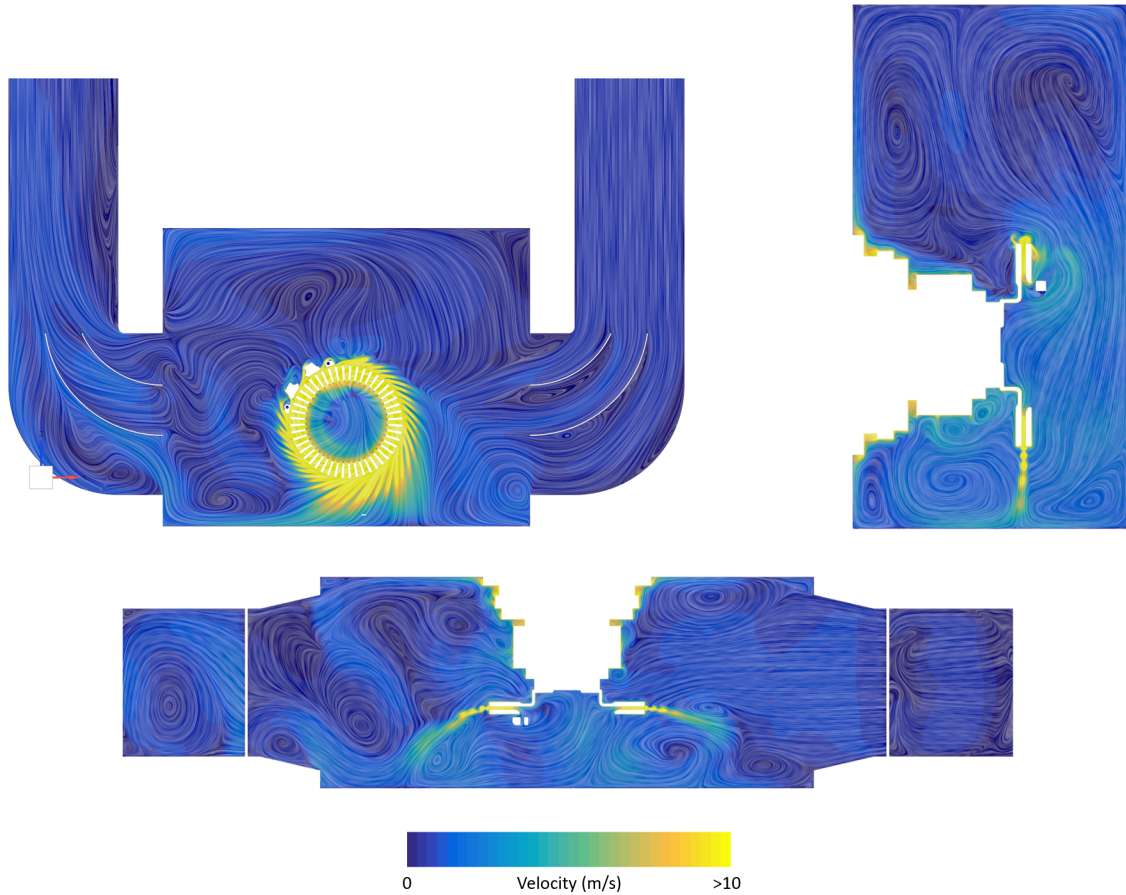


Figure 6.7: Cross sections of the computational Domain with the baseline configuration (500 °C, 180 km/h, 300 m³/h)

Analyzing the velocity and temperature fields across all three principal planes provides a comprehensive explanation for this hierarchical performance:

- **X-plane (Figure 6.7 top left):** This cross-section highlights the dominance

of heat dissipation by the ventilation cavity through the accelerated flow. Conversely, this view also reveals vortices and wake flows near the outlet regions, indicating that these surfaces are largely shielded from the ambient flow required for effective cooling.

- **Y-plane (Figure 6.7 top right):** This perspective illustrates the contrasting flow regimes between the front (inner) side and the mounting (outer) side of the disc. While the inner friction surface is consistently exposed to a steady, high-velocity inlet stream, the outer friction surface, outer swan neck, and outer hub interface are subjected to turbulence and recirculation zones. These stagnant pockets significantly reduce convective efficiency relative to the inner surfaces.
- **Z-plane (Figure 6.7 bottom):** This plane captures the internal recirculation within the disc's center and the aerodynamic interaction at the radial outer surfaces. The radial outer surface is effectively cooled at the inlet side but remains trapped in a wake zone at the outlet side. Furthermore, the geometric shielding of the hub and swan neck leads to lower velocities and stagnant air pockets. This lack of fluid exchange confirms why these features contribute minimally to the total cooling capacity, placing them at the base of the hierarchy.

By considering these three perspectives, it becomes clear that the hierarchical order of heat dissipation is a direct result of the local air velocity. The contributions of individual surfaces are visualized in the pie chart in Figure 6.8, highlighting which regions are most critical for the heat transfer from the disc.

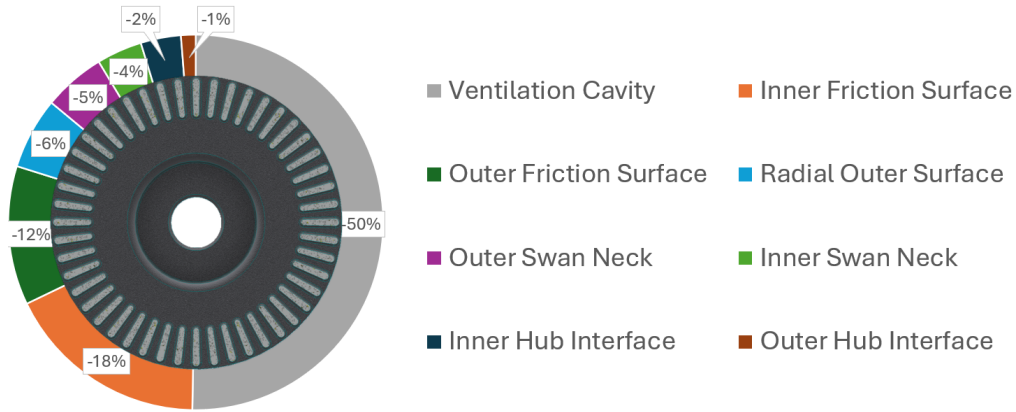


Figure 6.8: Heat transfer distribution (180 km/h, 300 m³/h)

While the general trends of total heat transfer and heat flux align, the hierarchical order of HTC's across specific surfaces differs due to variations in surface area. Because these metrics are area-averaged, small surfaces can yield disproportionately high values that do not reflect their true macro-cooling capacity, as governed by $\dot{Q} = hA\Delta T$. For instance, while the ventilation cavity provides the highest total heat transfer, its vast surface area causes its area-averaged heat flux to appear lower than that of smaller geometric features. Conversely, the outer hub interface exhibits nonphysically high HTC values that exceed even the primary cooling surfaces; this

artifact occurs because its effective area (indicated by the yellow surface in Figure 6.9 left) was drastically reduced by a Boolean subtraction operation during CAD assembly, as depicted in Figure 6.9 right. Additionally, this localized surface region may contain a small concentration of skewed grid cells characterized by localized, nonphysically high HTC values, which, when integrated into the surface average, artificially skew the overall metric upward.

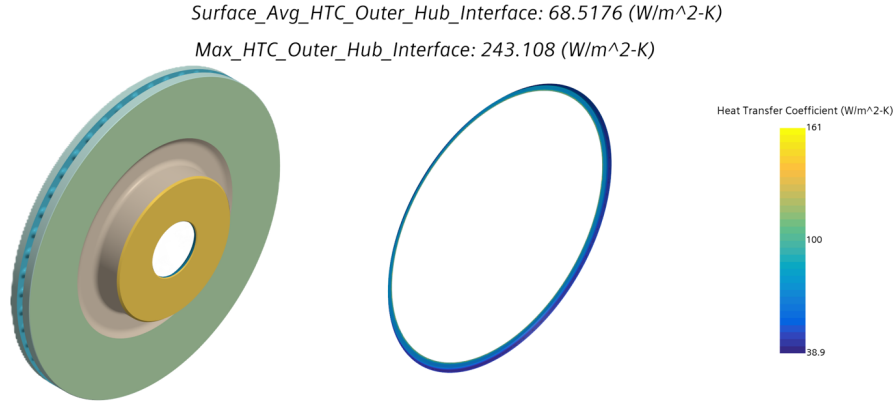


Figure 6.9: HTC of Outer Hub Interface (500 °C, 180 km/h, 300 m³/h)

In reality, this surface experiences no convective cooling because it is clamped in direct contact with the test-bench chuck, dissipating heat purely via solid-body conduction. This confirms that its high HTC is a numerical artifact of the area-averaging process rather than an indication of significant cooling performance. Consequently, to ensure an easier interpretation, the subsequent analysis focuses exclusively on the primary cooling surfaces (which drives approximately 75% of the disc's total cooling): the inner and outer friction surfaces, the ventilation cavity, and the radial outer surface only when notable aerodynamic behaviors are observed.

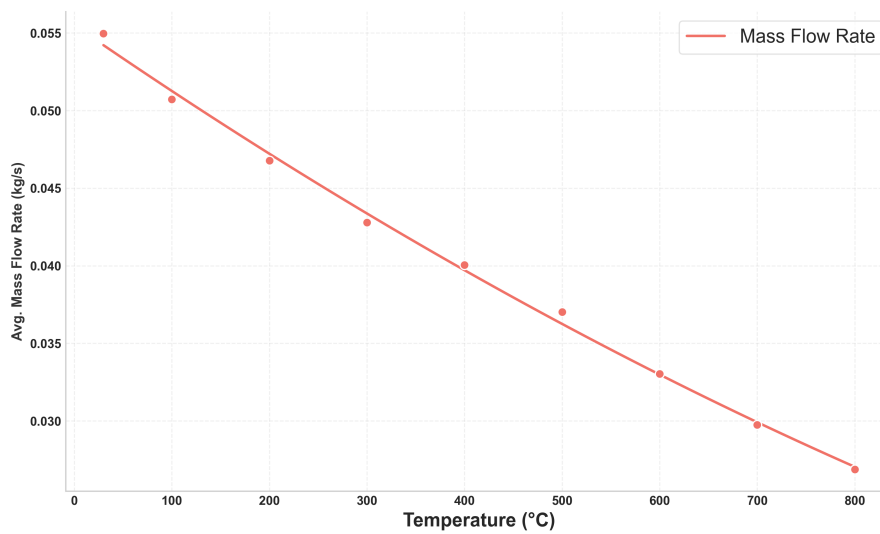


Figure 6.10: Averaged mass flow rate in the ventilation cavity (180 km/h, 300 m³/h)

Considering the mass flow rate (Figure 6.10) through the ventilation cavity, it decreases as surface temperature rises for a given constant rotational speed. This is governed by the previously mentioned decrease in density and viscosity. According to the continuity equation ($\dot{m} = \rho \dot{V}$), since the volumetric flow ($\dot{V}$) is mechanically driven by the disc's RPM and remains stable, the mass flow must decrease to satisfy mass balance [14].

6.3 Evaluation of HTC at Variable Rotational Speed

The parametric data presented in Figure 6.11 indicate a consistent decrease in the Heat Transfer Coefficient (HTC) as the brake surface temperature rises, though the magnitude of this degradation depends heavily on the velocity regime.

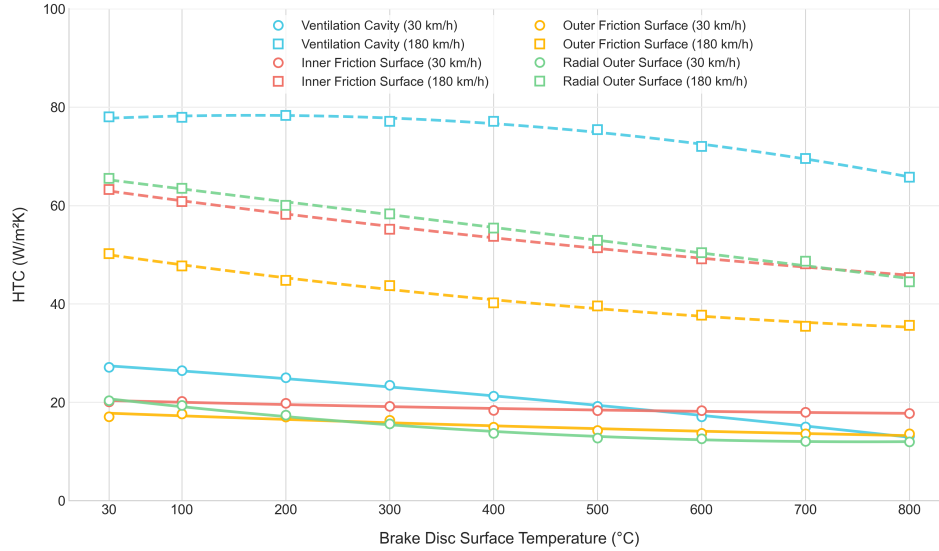


Figure 6.11: Surface averaged HTC as a function of surface temperature at 30 km/h and 180 km/h (500 °C, 300 m³/h)

Effect on internal vanes

At lower vehicle speeds, such as 30 km/h, the ventilation cavity's HTC drops sharply with temperature to nearly half its initial value, ultimately falling below that of the inner friction surface. This behavior demonstrates that temperature-induced flow resistance, or viscous damping, is highly dominant at lower velocities where the centrifugal forces are weak. Conversely, at a high operational speed of 180 km/h, the cavity's HTC decreases by $\approx 17\%$ across the same temperature range. The higher rotational speed amplifies the centrifugal effect sufficiently, as demonstrated by the velocity profiles and mass flow rates in Figure 6.12, to overcome the rising viscous shear forces and maintain core cooling performance.

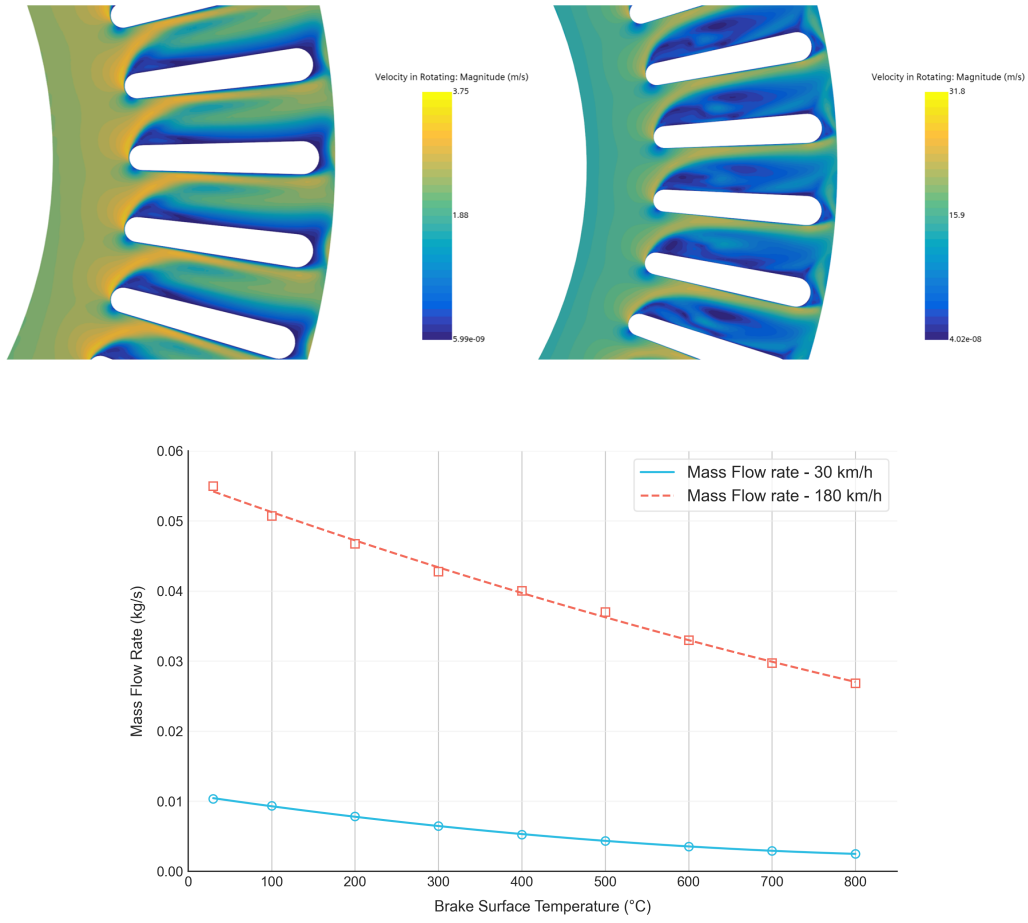


Figure 6.12: Averaged flow fields for 30 km/h and 180 km/h (500 °C, 300 m³/h)

Effect on external surfaces

Conversely, convective heat transfer on the external surfaces is primarily governed by their shearing action against the incoming airflow. At 30 km/h, this shearing mechanism is weak, causing the HTC to remain nearly constant or decrease only marginally as temperature rises. In contrast, thermal degradation of the HTC is far more pronounced at 180 km/h. The rapid shearing at high speeds significantly thins the boundary layer, making localized convective transport highly sensitive to temperature-dependent variations in fluid properties.

As opposed to the plot 6.11, which demonstrates the HTC across the temperature change, the direct influence of rotational speed on this heat dissipation is quantitatively evidenced by Figures 6.13, which exhibit a nearly linear relationship in HTC of the internal vanes and friction surfaces relative to rotational speed. This linear relationship aligns well with findings reported in the literature under comparable experimental conditions [22, 23]. Quantitatively, the HTC values elevate by a factor of approximately three to four as the vehicle speed scales from 30 km/h to 280 km/h. Even at the standard test procedure speed of 180 km/h, the HTC remains two to three times higher than the low-speed baseline, reinforcing the dominant role of rotational speed over the system's thermal performance.

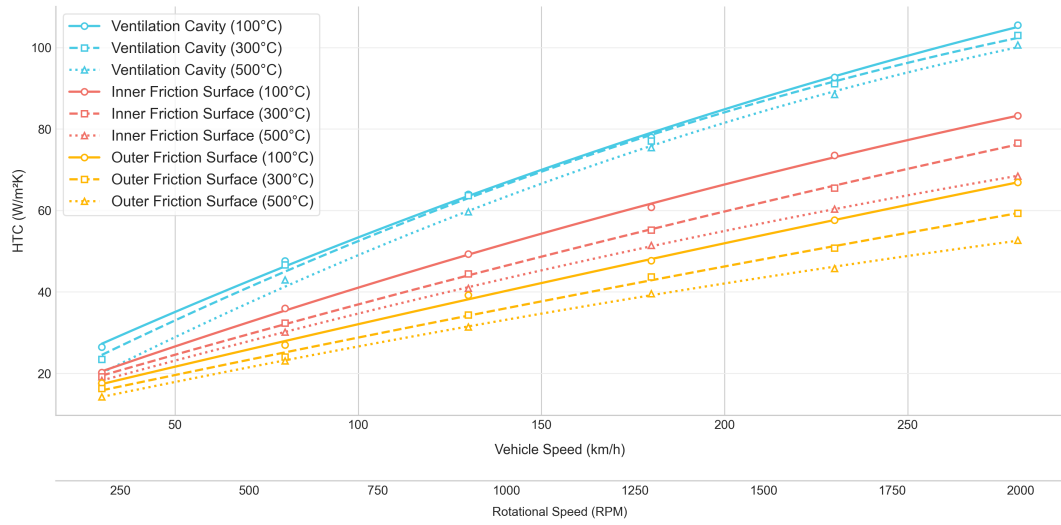


Figure 6.13: Surface averaged HTC as a function of rotational speeds at 100°C, 300°C and 500°C (180 km/h, 300 m³/h)

Because the HTC depends simultaneously on both temperature and rotational speed, this relationship can be effectively visualized as a three-dimensional surface plot. Furthermore, such an empirical representation facilitates the computationally efficient extraction of the HTC for any specific functional domain at a given temperature and velocity regime. This response surface is exemplified for the ventilation cavity alone in Figure 6.14.

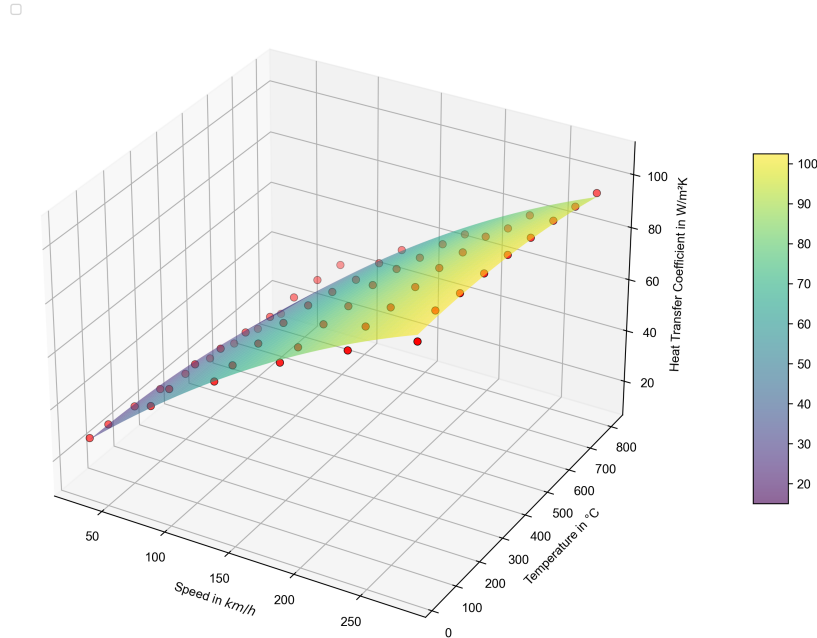


Figure 6.14: Surface plot for averaged HTC of ventilation cavity as a function of rotational speed and temperature

The generalized HTC regression equation corresponding to this plot is formulated as a function of surface temperature and rotational speed in Equation 6.1. Given that the localized HTC exhibit a non-linear degradation with respect to surface temperature alongside a linear-to-second-order progression with respect to rotational speed, a second-order polynomial surface fit was implemented:

$$h_S(T_S, v_{veh}) = \beta_0 + \beta_1 \cdot v_{veh} + \beta_2 \cdot T_S + \beta_3 \cdot v_{veh}^2 + \beta_4 \cdot T_S^2 + \beta_5 \cdot (v_{veh} \cdot T_S) \quad (6.1)$$

Where h_S represents the computed surface-averaged localized HTC of a specific surface, and β_0 through β_5 are the surface-specific regression coefficients compiled in the Appendix. Because convective heat transfer is fundamentally governed by local fluid properties and shearing mechanisms characterized by the dimensionless Reynolds and Prandtl numbers, an interaction term ($v_{veh} \cdot T_S$) is physically necessitated. This term mathematically captures the coupled effect of temperature-dependent thermo-physical property variations and high-speed shearing within the boundary layer.

6.4 Evaluation of HTC at Variable Rotational Direction

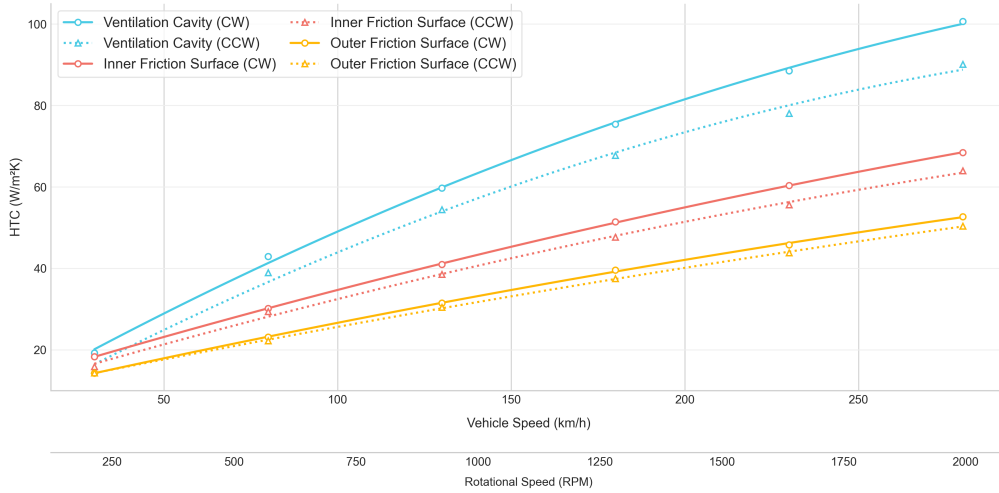


Figure 6.15: Surface averaged HTC as a function of rotational speed at clockwise (CW) and counterclockwise (CCW) rotational direction (500 °C, 180 km/h, 300 m³/h)

For identical rotational speeds in the opposite rotational direction, the results in Figure 6.15 follow an overall similar trend. However, the absolute values differ significantly. The heat transfer characteristics for counterclockwise rotation are lower than those observed for clockwise rotation. This discrepancy is likely rooted in the complex viscous interaction between the disc surface and the ambient airflow.

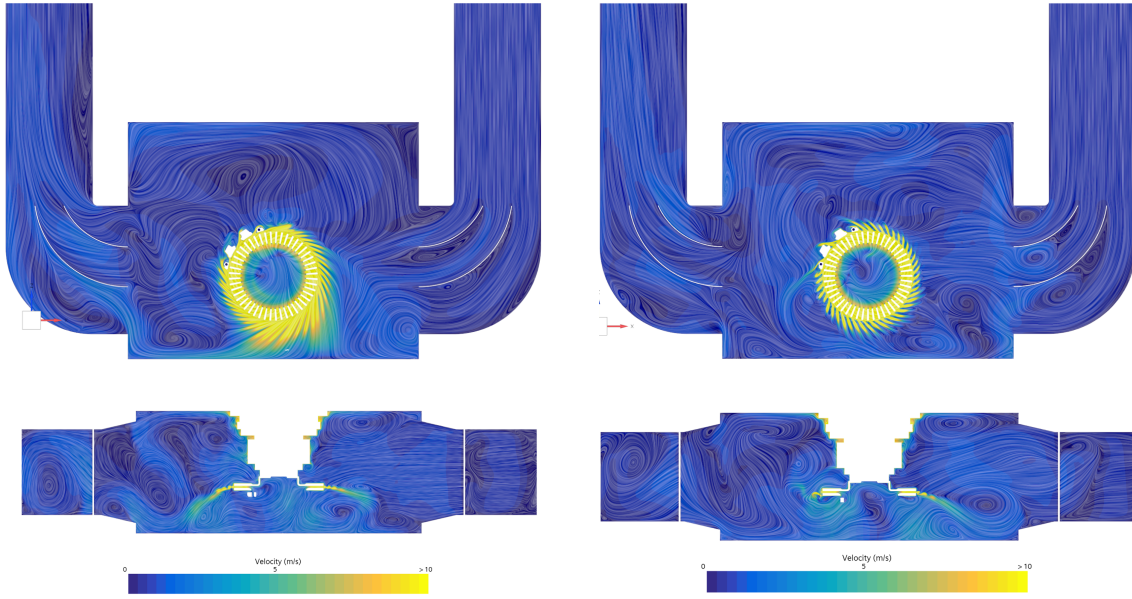


Figure 6.16: Velocity plot for clockwise (CW) and counterclockwise (CCW) rotation (500 °C, 180 km/h, 300 m³/h). Top left: x-plane CW, top right: x-plane CCW, bottom left: z-plane CW, bottom right: z-plane CCW

The flow dynamics associated with rotational direction are visually clarified through the comparative visualizations in Figure 6.16. In the clockwise configuration (top left of Figure 6.16), the disc surface rotates co-directionally with the incoming air-flow, promoting a stable boundary layer with minimal flow separation and low recirculation. This prolonged contact with the cool freestream enhances convective heat transfer. Conversely, counterclockwise rotation (top right) forces the disc surface directly against the incoming flow. This counter-directional interaction creates high-shear gradients that introduce recirculation zones and turbulent eddies from the dead wake zone at the leading edge. These eddies act as a thermal barrier by trapping and decelerating air near the disc, insulating the hot surfaces and substantially inhibiting convective efficiency.

6.5 Evaluation of HTC at Variable Inlet Airflow Rate

The influence of airflow rate on convective performance is analyzed by systematically varying the volumetric air inlet to the test-bench. As illustrated by the parametric data in Figure 6.17, increasing the inlet airflow yields distinct, non-uniform enhancements across the disc's functional surfaces. While the HTC of the inner friction surface exhibits only a marginal upward shift, the HTCs of both the ventilation cavity and the outer friction surface display a clear, nearly linear correlation with the rising volumetric airflow rate.

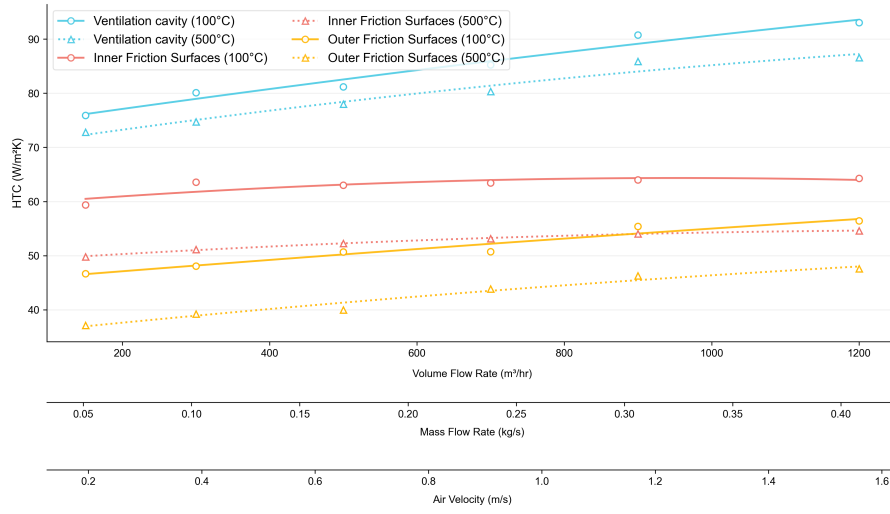


Figure 6.17: Surface averaged HTC as a function of test-bench inlet airflow rates (500 °C, 180 km/h)

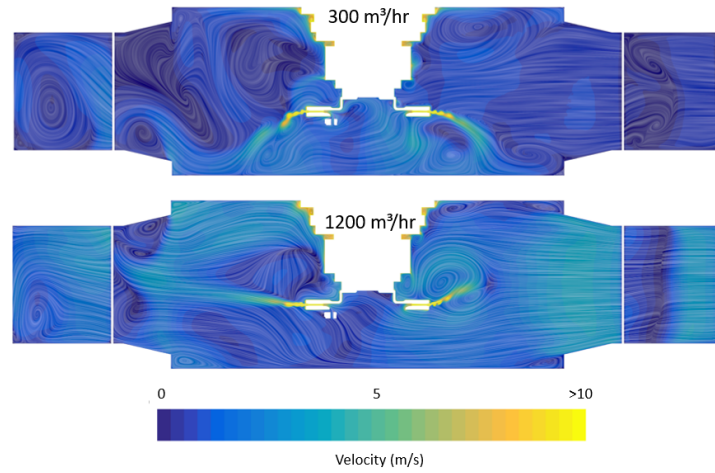


Figure 6.18: Velocity plot for different test-bench inlet airflow rates (500 °C, 180 km/h). Top: 300 m³/h, bottom: 1200 m³/h)

Cross-sectional analysis at the z-plane (Figure 6.18) clarifies the flow physics driving these non-uniform cooling enhancements. At a lower inlet airflow rate of 300 m³/h, the weak stream only slightly deflects the internal vane exhaust, allowing it to maintain cooling contact with the inner surface, while the outer friction surface remains trapped in an aerodynamic shadow region hindered by vortices. Conversely, at 1200 m³/h, the high-velocity inlet stream is strong enough to sharply deflect this vane exhaust directly toward the outer friction surface, significantly supplementing its heat dissipation. Because the inner surface maintains constant exposure to the freestream while the outer surface is aerodynamically shielded under low-flow conditions, the inner friction surface consistently maintains a higher HTC and absolute heat transfer rate across all tested configurations.

6.6 Evaluation of HTC for Variable Brake Design

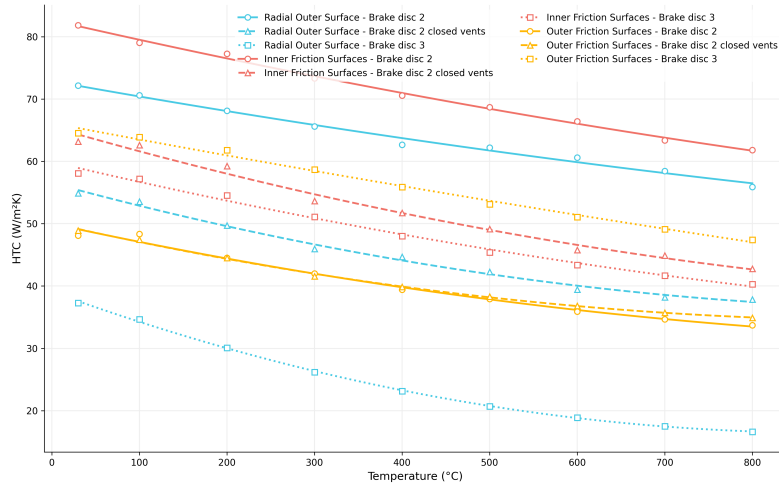


Figure 6.19: Comparison of HTC of friction and radial surface for brake disc 2 vs brake disc 2 with closed vents vs solid brake disc

A comparative evaluation of the HTC for the baseline ventilated Brake Disc 2, Brake Disc 2 with closed vanes, and the solid Brake Disc 3 is presented in Figure 6.19. The profiles reveal distinct structural variations between the ventilated and solid configurations. Interestingly, Brake Disc 2 maintains the same hierarchical order of surface HTC whether its internal cooling channels are open or closed, whereas the solid Brake Disc 3 exhibits a completely different sequence. To determine if this behavior is driven by altered aerodynamics or surface area asymmetry, specifically, the solid disc's smaller outer (0.04 m^2) versus inner (0.05 m^2) friction surface area, the absolute heat transfer is plotted in Figure 6.21.

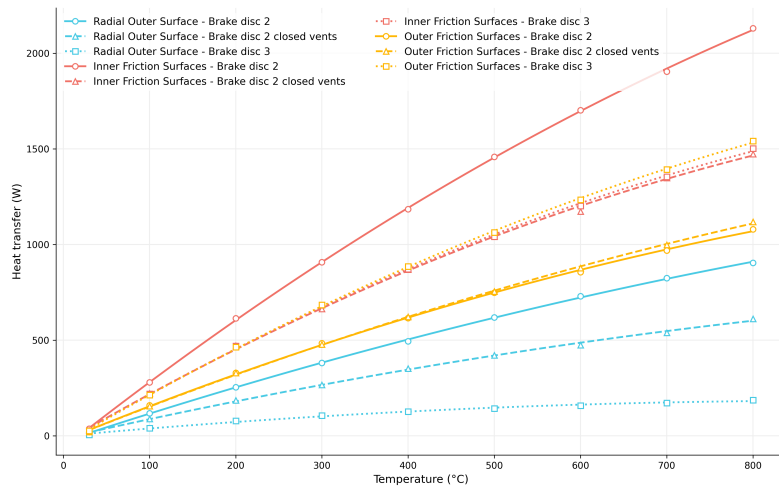


Figure 6.20: Comparison of Heat transfer of friction and radial surface for brake disc 2 vs brake disc 2 with closed vents vs solid brake disc

Brake Disc 2 open vs closed vanes

Isolating the internal flow paths within the same primary geometry reveals that closing the ventilation cavity yields negligible changes along the main friction faces, implying that the external surfaces remain largely unaffected. In contrast, the radial outer surface suffers a severe reduction in absolute heat transfer when the internal vanes are closed.

In the baseline ventilated configuration, the centrifugal acceleration of the internal air mass ejects high-velocity jets directly near the radial outer boundary layer. When the channels are closed, this localized velocity boost is entirely lost. The oncoming external airflow stalls against the radial boundary, creating a stagnation zone that severely limits convective dissipation. This can be visualized in Figure 6.21.

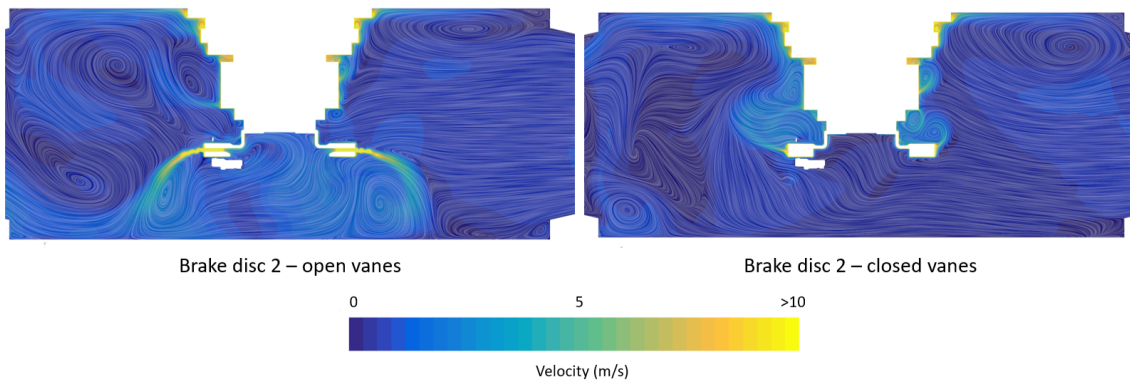


Figure 6.21: Comparison of velocity at z-plane for brake disc 2 with open and closed vanes

Brake disc 2 vs brake disc 3

Comparing the ventilated brake disc 2 to the solid Brake Disc 3 reveals that while the solid configuration provides lower overall cooling performance, its primary friction surfaces contribute equally to the thermal dissipation. Beyond the geometric changes, this behavior may also be influenced by the usage of a different setup, especially the caliper assembly. The brake disc 3 utilizes a different brake caliper compared to the one paired with the ventilated Brake Disc 2.

Despite these structural modifications, the hierarchical sequence of total energy dissipation remains consistent with the baseline, with the radial outer surface contributing the least to total cooling. However, due to the complete removal of the internal cavity, the radial boundary of the solid disc exhibits a critical drop in absolute heat transfer compared to both variations of Brake Disc 2. This deviation is a result of a reduction in the surface area of the radial outer surface as a consequence of the absence of the vane cavity. To visualize how these geometric and aerodynamic variations alter the global thermal balance, the proportional heat dissipation breakdowns are illustrated via the distribution charts in Figure 6.22. When comparing the two configurations with identical external surface areas (Brake Disc 2 with open vs. closed vanes), blocking the internal cavity causes global convective heat dissipation to collapse by 56.8%, dropping from 7617.12 W to 3287.61 W as the inner and

outer friction surfaces scale up to absorb 23% and 32% of the remaining thermal load.

Interestingly, the solid Brake Disc 3 achieves a virtually identical total heat transfer rate (3291.28 W) to the closed-vane variant, despite its drastically reduced thickness and profile mass. However, their internal distributions differ significantly: the reduced thickness of the solid disc allows its inner and outer friction surfaces to dissipate a perfectly balanced, symmetrical share (32% each) of the total convective load, whereas the closed-vane design remains thermally asymmetrical due to its unaltered, wider body casting.

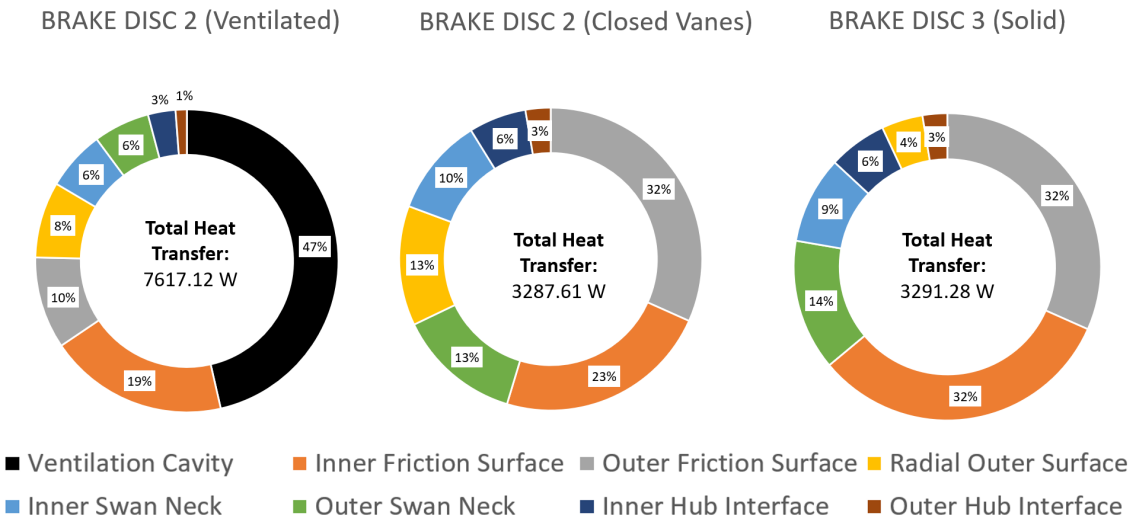


Figure 6.22: Comparison of Heat distribution by the surfaces of the ventilated brake disc (Brake disc 2) and solid brake disc (Brake disc 3)

7

Experimental Validation

This section evaluates how the CFD-extracted Heat Transfer Coefficients (HTCs) correlate with physical data from a dynamometer test-bench. Because this validation is a minor part of the thesis scope, the focus is placed on the correlation of temperature from CFD/FEM Simulation data and dyno data. The test-bench maintains a constant duct airflow rate of $1200 \text{ m}^3/\text{h}$ for this test procedure. Continuous temperature data is captured via an embedded thermocouple on the friction plane between two vanes, illustrated using Figure 7.1. For an example case, the validation is based on a sequence-stopping test consisting of 15 consecutive braking and cooling cycles, one of which is illustrated in Figures 7.2 and 7.3.

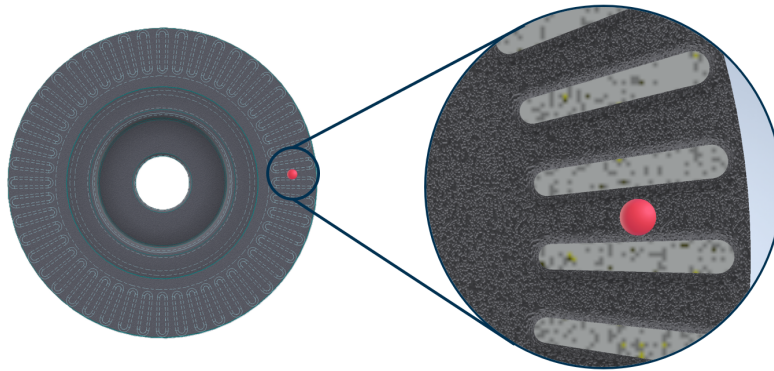


Figure 7.1: Schematic visualization of the location of the thermocouple on a ventilated brake disc indicated by the red dot

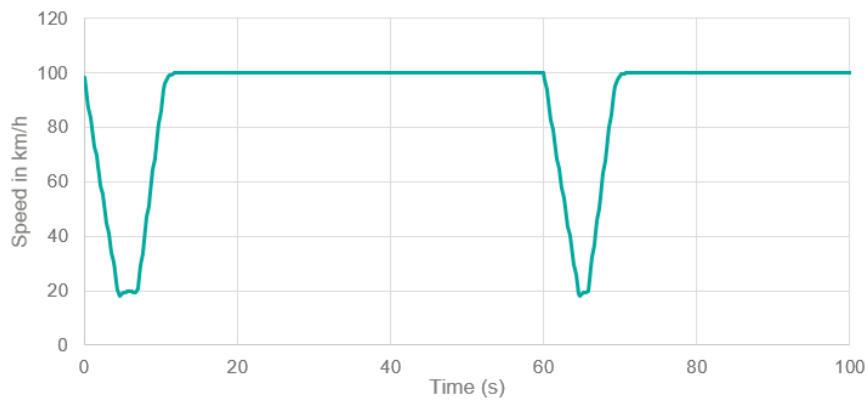


Figure 7.2: Speed of a single braking cycle from one of the test cases

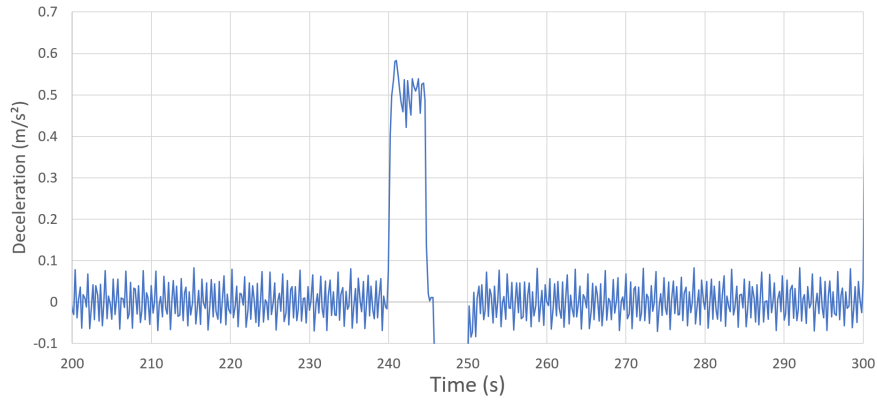


Figure 7.3: Deceleration of a single braking cycle from one of the test cases

The structural thermal response is solved using a transient heat transfer model in Abaqus. To minimize computational expense, a one-quarter symmetric sector of the disc is utilized, as shown in Figure 7.4. As mentioned in section 2.8, the brake disc needs all three heat transfer mechanisms to dissipate heat. While the CFD domain was restricted to convection, the FEM model incorporates solid-body conduction and surface radiation. Conduction is defined in terms of temperature-dependent material properties: thermal conductivity, density, and specific heat capacity.

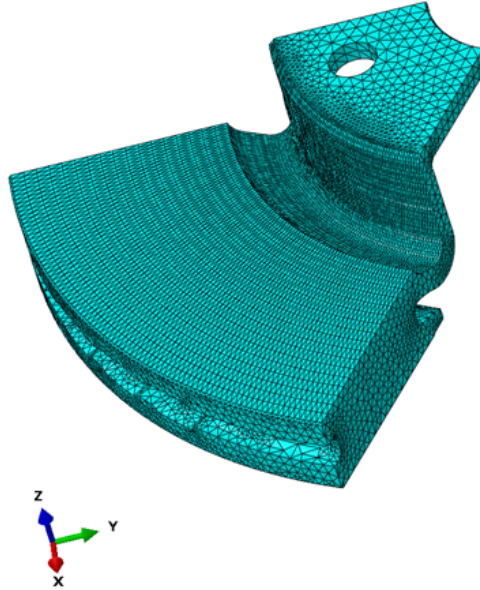


Figure 7.4: Symmetric sliced part of the brake disc used for FEM simulations

The geometry is discretized with approximately 135,000 high-order DC3D10 elements (10-node quadratic heat transfer tetrahedra). To replicate post-bedding laboratory conditions and since the test procedure has a start temperature, a uniform initial temperature of 50°C is applied across the domain as a predefined field. Mechanical friction is modeled directly as an influx of thermal energy. A heat flux equation, based on the formulation by Belhocine et al.[24], is applied homogeneously across the friction surfaces during each braking phase, as shown in Figure 7.5.

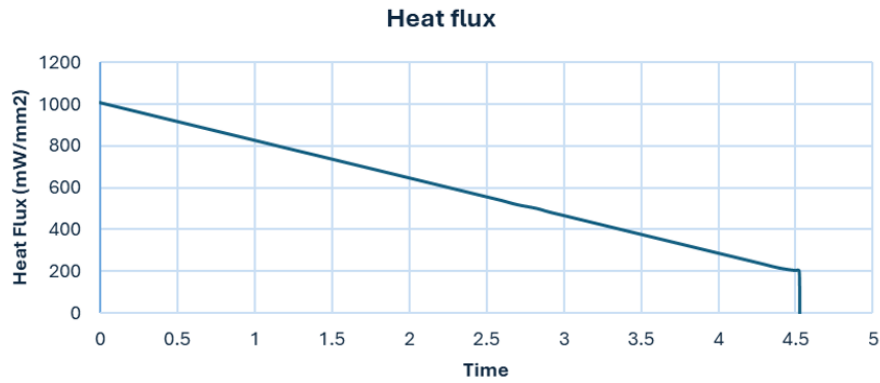


Figure 7.5: Heat flux curve for mechanical braking

For the validation test (designated as 'CAE_run01' in Figure 7.6), the transient thermal solver in Abaqus was executed by incorporating the CFD-derived convective HTC's alongside radiation and conduction. Radiation was modeled using a simplified constant emissivity of 0.15 to approximate the highly transient surface conditions, while physical assembly boundaries, such as the outer hub interfaces clamped against the test-bench chuck, were modeled via an equivalent conduction boundary condition to replicate heat dissipation into the chuck. Once all material properties, mesh parameters, and heat flux profiles were integrated, the resulting numerical temperature data were extracted from the friction plane at a location corresponding to the embedded physical thermocouple and plotted directly against the raw dynamometer datasets in Figure 7.6 to evaluate the correlation.

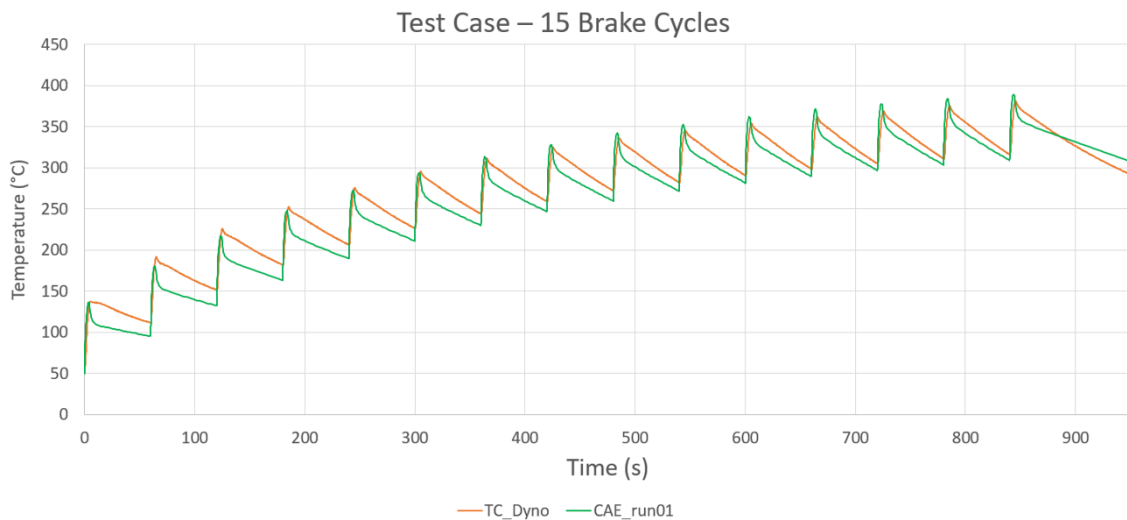


Figure 7.6: Sequence stop test with temperature from dyno and CFD/FEM simulations

In the transient temperature plot, the orange curve represents the experimental thermocouple data, while the green curve corresponds to the numerical simulation results. The convective HTC's extracted from the CFD domain, when coupled with conduction at the outer hub interface and a surface emissivity of 0.15, yield a thermal

response that closely correlates with the physical dataset. Although the numerical model exhibits a deviation during the cooling phases, which is more pronounced during the initial cycles and reduces in later stages, the peak temperatures captured during the braking events remain nearly identical, showing a deviation of only a few degrees. Consequently, these results demonstrate that the established CFD framework, combined with specific boundary conditions, successfully replicates the laboratory dynamometer environment.

8

Conclusions

This thesis investigated the localized thermal characteristics of automotive brake discs, specifically focusing on temperature-dependent HTC through a numerical methodology. The numerical framework, involving a steady-state RANS approach with $k - \omega$ Shear Stress Transport (SST) turbulence model and MRF rotational modeling, provides a reliable yet computationally efficient method for investigating the heat transfer characteristics.

A major methodological insight was that high-temperature convection simulations are highly sensitive to fluid property definitions. Approximating air with constant properties significantly under-predicts heat transfer, proving that convective dissipation is driven non-linearly by temperature-induced property variations within the film boundary layer, rather than a simple linear temperature differential (ΔT).

Baseline simulations revealed that localized surface-averaged HTCs steadily decline across most surfaces despite an increase in absolute heat flux and heat transfer. This degradation is driven by thermodynamic shifts at elevated film temperatures—specifically, a severe drop in fluid density and an increase in dynamic viscosity. Inside the ventilation cavity, these changes reduce the mass flow rate and cause a significant drop in vane outlet velocity. On external surfaces, the rising kinematic viscosity thickens the boundary layer, increasing thermal resistance and inhibiting convective dissipation.

Parametric variations showed that increasing rotational speed yields a linear enhancement in global convective cooling dominated by the centrifugal pumping forces of the internal ventilation cavity, while external friction faces experience minor gains via boundary layer thinning. Furthermore, reversing the rotational direction reduces average HTCs due to severe flow recirculation zones from a "dead wake" area against the oncoming airflow.

Altering the test-bench inlet airflow flow rate optimized cooling only selectively. Internal cavity HTCs remained less sensitive to external flow changes because internal mass flow is governed entirely by the rotor's independent centrifugal pumping, yet it still showed measurable linear HTC gain. However, the external friction surfaces and flow-restricted zones, previously suppressed by a localized deficit of cooling air shadowed zones, utilized this increased mass flow to enhance their convective dissipation and exhibit a linear cooling increase.

The structural sensitivity study demonstrated the geometric importance of internal ventilation channels. Closing the internal cavity resulted in a massive performance drop, causing global convective dissipation to decrease to nearly half ($\approx 44\%$) of

its baseline capacity. While the internal core absorbs the largest share of the thermal load in the ventilated design (47%), a solid geometry forces a significant heat redistribution, forcing the primary inner and outer friction surfaces to assume a symmetrical proportion (32% each) of the convective load.

Finally, the CFD-extracted HTC's were successfully validated through transient FEM simulations mapped against physical dynamometer test-bench data, demonstrating that the coupled CFD-FEM computational model is fully capable of replicating experimental datasets.

8.1 Open Points

This study serves as a foundational numerical baseline for Volvo Cars Propulsion Components by establishing a temperature-dependent convective HTC model, opening up opportunities to expand the project across wider testing conditions, material configurations, and geometric designs.

Due to time constraints, the investigation was restricted to a steady-state thermal framework that omitted solid material properties. A natural next step is to transition to fully transient heat transfer analyses that incorporate material properties alongside coupled conduction and radiation, while utilizing a Rigid Body Motion (RBM) sliding mesh model to capture transient rotational effects on the localized HTC [13]. This could streamline the workflow and drastically reduce total data-mapping and engineering time.

To refine the computational domain, geometric simplifications like the duct deflector vanes and mounting chuck should be replaced with exact manufacturing CAD models, while the chuck could be modeled using experimentally logged transient temperatures rather than a constant 20°C boundary condition. Although omitting the auxiliary caliper fixture maintained a standardized comparison across both brake discs, future models should include it, as its structural arms protrude into the hub cavity to generate localized turbulence that alters hub cooling.

As a final step, test-bench-derived HTC's can be correlated with full-vehicle computational domains to investigate how the enclosed-wheel environment and rotating wheel rims alter the cooling behavior compared with laboratory conditions.

Bibliography

- [1] R. Limpert, Brake Design and Safety, 3rd Edition, 2011.
- [2] UNECE Brake System legislative Regulations, Link: World Forum for Harmonization of Vehicle Regulations.
- [3] R. Abbasi, Design of Machine Tool for Motor Cycle Floating Disc Brakes, 2018.
- [4] M. Liermann, C. Stammen, H. Murrenhoff, Development of a Self-Energizing Electro-Hydraulic Brake (SEHB), 2007, SAE Technical Papers, DOI: <https://doi.org/10.4271/2007-01-4236>.
- [5] Solid or Vented, retrieved on February 02, 2026, from https://www.mtecbrakes.com/solid_or_vented.
- [6] Drilled vs slotted: Which brake disc to choose?, retrieved on February 02, 2026 from <https://www.brembo.com/en/news-archive/disc-drilled-or-slotted>.
- [7] B. Breuer, K.H. Bill, Bremsenhandbuch, 5th Edition, 2017.
- [8] Link Equipment - Model 3000 Performance Brake Dynamometer, retrieved on May 20, 2026, from <https://www.linkeng.com/product/model-3000-performance-brake-dynamometer/>.
- [9] R. Schwarze, CFD-Modellierung: Grundlagen und Anwendungen bei Strömungsprozessen, 1st Edition, 2013.
- [10] B.R. Munson, T.H. Okiishi, W.W. Huebsch, A.P. Rothmayer, Fundamentals of Fluid Mechanics, 7th Edition, 2013.
- [11] A.Y. Cengel, Heat Transfer: A Practical Approach, 2nd Edition, 2002
- [12] T. Hobeika, S. Sebben, CFD investigation on wheel rotation modelling, 2018, DOI: <https://doi.org/10.1016/j.jweia.2018.01.005>.
- [13] T. Dekker, V. Krishnan, Numerical Investigations of Brake Cooling Performance, 2018.

- [14] T.K. Sathyan, A. Vdovin, Experimental thermal analysis of brake cooling relative to mass flow of air through a ventilated brake disc, *Case Studies in Thermal Engineering*, Volume 64, 2024, DOI: <https://doi.org/10.1016/j.csite.2024.105530>.
- [15] D. Parish, D.G. MacManus, Aerodynamic investigations of ventilated brake discs, 2004, DOI: <https://doi.org/10.1243/095440705X11121>.
- [16] W. Wang, Y. Cao, T. Okaze, Comparison of hexahedral, tetrahedral and polyhedral cells for reproducing the wind field around an isolated building by LES, 2021, DOI: <https://doi.org/10.1016/j.buildenv.2021.107717>.
- [17] A. Jerhamre, C. Bergström, Numerical Study of Brake Disc Cooling Accounting for Both Aerodynamic Drag Force and Cooling Efficiency, 2001, SAE 2001 World Congress of Mechanical Engineering Science, DOI: <https://doi.org/10.4271/2001-01-0948>.
- [18] S.M. Reddy, J.M. Mallikarjuna, V. Ganesan, Flow and Heat Transfer Analysis Through a Brake Disc: A CFD Approach, 2006, ASME International Mechanical Engineering Congress and Exposition, DOI: <https://doi.org/10.1115/IMECE2006-14317>.
- [19] N.P. Gulhane, S.P. Mahulikar, Numerical study of compressible convective heat transfer with variations in all fluid properties, *International Journal of Thermal Sciences*, Volume 49, Issue 5, 2010, DOI: <https://doi.org/10.1016/j.ijthermalsci.2009.11.001>.
- [20] N.J. Greiner, M.D. Polanka, J.L. Rutledge, J.R. Robertson, Effect of Variable Properties Within a Boundary Layer With Large Freestream-to-Wall Temperature Differences, 2014, *Journal of Engineering for Gas Turbines and Power*, Volume 136, Issue 5, DOI: <https://doi.org/10.1115/1.4026117>.
- [21] J. Zhao, J. Zhu, Y. Xu, Y. Li, New generalized expressions for forced convective heat transfer coefficients across the flat plate, 2025, *Applied Thermal Engineering*, Volume 258, Part B, Article: 124688, DOI: <https://doi.org/10.1016/j.applthermaleng.2024.124688>.
- [22] T. Çakmak, G. Sevilgen, M. Kılıç, Heat Transfer Enhancement in Dry Clutch Pressure Plate Using Ventilation Channels, 9th International Automotive Technologies Congress, OTEKON, 2018, retrieved on 27th April 2026, Link.
- [23] A.D. McPhee, D. Johnson, Experimental heat transfer and flow analysis of a vented brake rotor, *International Journal of Thermal Sciences*, April 2008, DOI: <https://doi.org/10.1016/j.ijthermalsci.2007.03.006>.

- [24] A. Belhocine, A.R. Abu Bakar, M. Bouchetara, Numerical Modeling of Disc Brake System in Frictional Contact, *Tribology in Industry*, Volume 35, March 2014, retrieved on 27th April 2026 from <https://www.tribology.rs/journals/2014/2014-1/7.pdf>.

A

Appendix 1

Appendix 1 consists of the raw data points corresponding to the experimental/simulation results presented in Section 6.

For clarity, the abbreviations in the matrix represent the following monitored surfaces: Ventilation Cavity (VC), Inner Friction Surface (IFS), Outer Friction Surface (OFS), Inner Swan Neck (ISN), Outer Swan Node (OSN), Inner Hub Interface(IHI), Outer Hub Interface (OHI), and Radial Outer Surface (ROS).

Table A.1: Surface-averaged HTC values ($\text{W}/\text{m}^2\text{K}$) for the baseline configuration (Brake disc 1) at various temperatures ($^{\circ}\text{C}$) (180 km/h, 300 m^3/h)

Temperature	VC	IFS	OFS	ISN	OSN	IHI	OHI	ROS
30	78.03	63.25	42.04	44.11	50.24	66.32	36.88	65.49
100	77.93	60.81	38.73	39.89	47.71	68.33	35.27	63.50
200	78.33	58.22	36.32	37.82	44.80	67.51	33.37	60.02
300	77.09	55.19	37.14	37.70	43.70	70.88	32.01	58.31
400	77.12	53.71	35.68	35.63	40.23	66.79	29.45	55.43
500	75.42	51.44	33.48	34.33	39.60	67.84	31.13	52.89
600	72.00	49.14	31.80	32.42	37.73	66.79	30.24	50.42
700	69.53	48.14	30.06	29.69	35.43	65.79	26.81	48.66
800	65.78	45.37	29.47	28.33	35.64	69.98	28.23	44.49

Table A.2: Surface-averaged heat flux values (W/m^2) for the baseline configuration (Brake disc 1) at various temperatures ($^{\circ}\text{C}$) (180 km/h, 300 m^3/h)

Temperature	VC	IFS	OFS	ISN	OSN	IHI	OHI	ROS
30	780.31	632.48	420.39	441.09	502.38	663.21	368.75	654.91
100	6234.22	4864.82	3098.11	3191.26	3816.77	5466.73	2821.36	5079.93
200	14099.38	10480.33	6537.08	6806.90	8063.93	12151.63	6006.29	10804.17
300	21583.95	15452.35	10399.15	10555.69	12236.23	19845.38	8962.21	16327.09
400	29305.40	20411.38	13558.73	13537.81	15288.16	25381.83	11191.75	21062.84
500	36201.74	24690.15	16070.84	16479.88	19008.46	32563.52	14942.96	25388.52
600	41759.19	28499.03	18444.49	18804.25	21882.38	38739.56	17536.87	29241.98
700	47279.92	32738.19	20441.10	20189.23	24093.85	44738.32	18231.20	33089.41
800	51305.40	35390.11	22985.54	22096.73	27800.16	54586.50	22021.75	34705.34

Table A.3: Surface-averaged heat transfer values (W) for the baseline configuration (brake disc 1) for surface temperatures ($^{\circ}\text{C}$) (180 km/h, 300 m^3/h)

Temperature	VC	IFS	OFS	ISN	OSN	IHI	OHI	ROS
30	98.74	38.98	6.33	15.70	29.33	1.94	12.60	15.34
100	788.87	299.83	46.68	113.60	222.81	16.02	96.40	118.98
200	1784.12	645.93	98.50	242.31	470.75	35.61	205.22	253.06
300	2731.21	952.37	156.70	375.75	714.32	58.17	306.21	382.42
400	3708.27	1258.00	204.31	481.91	892.49	74.40	382.39	493.34
500	4580.93	1521.71	242.17	586.64	1109.67	95.45	510.56	594.65
600	5284.16	1756.46	277.93	669.38	1277.44	113.55	599.18	684.91
700	5982.75	2017.73	308.02	718.68	1406.54	131.13	622.91	775.03
800	6492.13	2181.18	346.36	786.58	1622.91	159.99	752.42	812.88

Table A.4: Surface-averaged HTC values (W) for the baseline configuration (Brake disc 1) for surface temperatures ($^{\circ}\text{C}$) at 30km/h and 180 km/h (300 m³/h)

Temp	VC		IFS		OFS		ROS	
	30 kmph	180 kmph	30 kmph	180 kmph	30 kmph	180 kmph	30 kmph	180 kmph
30	27.11	78.03	20.12	63.25	17.07	50.24	20.37	65.49
100	26.46	77.93	20.19	60.81	17.67	47.71	19.37	63.50
200	25.03	78.33	19.80	58.22	17.03	44.80	17.39	60.02
300	23.47	77.09	19.19	55.19	16.35	43.70	15.64	58.31
400	21.24	77.12	18.39	53.71	14.97	40.23	13.69	55.43
500	19.21	75.42	18.33	51.44	14.28	39.60	12.77	52.89
600	17.09	72.00	18.32	49.14	13.72	37.73	12.63	50.42
700	15.01	69.53	17.99	48.14	13.59	35.43	12.08	48.66
800	13.19	65.78	17.73	45.37	13.61	35.64	12.00	44.49

Table A.5: Coefficients of generalized HTC equation for brake disc 1 (300 m³/h)

Surface Name	β_0	β_1	β_2	β_3	β_4	β_5
VC	17.9	$3.70e^{-1}$	$-1.047e^{-2}$	$-3e^{-4}$	$-1e^{-5}$	$5e^{-5}$
IFS	13.5	$2.52e^{-1}$	$-3.31e^{-3}$	$-3.2e^{-5}$	$-3e^{-6}$	$-2.1e^{-5}$
IHI	8.53	$1.87e^{-1}$	$-3.18e^{-3}$	$-7.8e^{-5}$	$1.5e^{-6}$	$-2.8e^{-5}$
ISN	8.07	$2.10e^{-1}$	$-3.92e^{-3}$	$-8.5e^{-5}$	$1.7e^{-6}$	$-2.4e^{-5}$
OFS	12.5	$2.17e^{-1}$	$-1.25e^{-2}$	$-1.05e^{-4}$	$9.6e^{-6}$	$-1.5e^{-5}$
OHI	17.9	$3.75e^{-1}$	$-2.82e^{-2}$	$-5.78e^{-4}$	$4.1e^{-6}$	$-1.8e^{-5}$
OSN	8.86	$1.70e^{-1}$	$-5.52e^{-2}$	$-1.35e^{-4}$	$3.5e^{-6}$	$-1.81e^{-5}$
ROS	13.5	$2.52e^{-1}$	$-3.34e^{-3}$	$-3.4e^{-5}$	$3e^{-6}$	$-2.1e^{-5}$

Table A.6: Surface-averaged HTC values (W) for the baseline configuration (Brake disc 1) at various temperatures ($^{\circ}\text{C}$) at 30km/h and 180 km/h (500 $^{\circ}\text{C}$, 300 m^3/h)

Speed (km/h)	VC		IFS		OFS	
	CW	CCW	CW	CCW	CW	CCW
30	19.21	14.97	18.33	15.87	14.28	14.33
80	42.95	38.96	30.14	29.37	23.12	22.23
130	59.70	54.40	40.96	38.52	31.45	30.44
180	75.42	67.77	51.44	47.69	39.60	37.46
230	88.48	78.05	60.34	55.58	45.81	43.83
280	100.65	90.14	68.45	64.00	52.70	50.39

Table A.7: Coefficients of generalized HTC equation for brake disc 2 (300 m^3/h)

Surface Name	β_0	β_1	β_2	β_3	β_4	β_5
VC	17.9	$3.70e^{-1}$	$-1.045e^{-2}$	$-3.02e^{-4}$	$-1.1e^{-5}$	$4.9e^{-5}$
IFS	18.7	$3.80e^{-1}$	$-1.245e^{-2}$	$-3.75e^{-4}$	$4.5e^{-6}$	$-1.5e^{-5}$
IHI	8.42	$1.784e^{-1}$	$-5.12e^{-3}$	$-1.15e^{-4}$	$3.1e^{-6}$	$-1.4e^{-5}$
ISN	7.54	$1.945e^{-1}$	$-4.12e^{-3}$	$-8.8e^{-5}$	$2.1e^{-6}$	$-1.8e^{-5}$
OFS	13.5	$2.11e^{-1}$	$-1.214e^{-2}$	$-1.01e^{-4}$	$8.9e^{-6}$	$-1.6e^{-5}$
OHI	17.4	$3.64e^{-1}$	$3.12e^{-3}$	$-5.48e^{-4}$	$3.8e^{-6}$	$-1.6e^{-5}$
OSN	10.1	$1.74e^{-1}$	$-5.12e^{-3}$	$-1.32e^{-4}$	$3.1e^{-6}$	$-1e^{-5}$
ROS	17.9	$3.7e^{-1}$	$-1.05e^{-2}$	$-3e^{-4}$	$-1e^{-5}$	$5e^{-5}$

B

Appendix 2

Appendix 2 consists of the images and plots corresponding to the experimental/simulation results presented in Section 6.

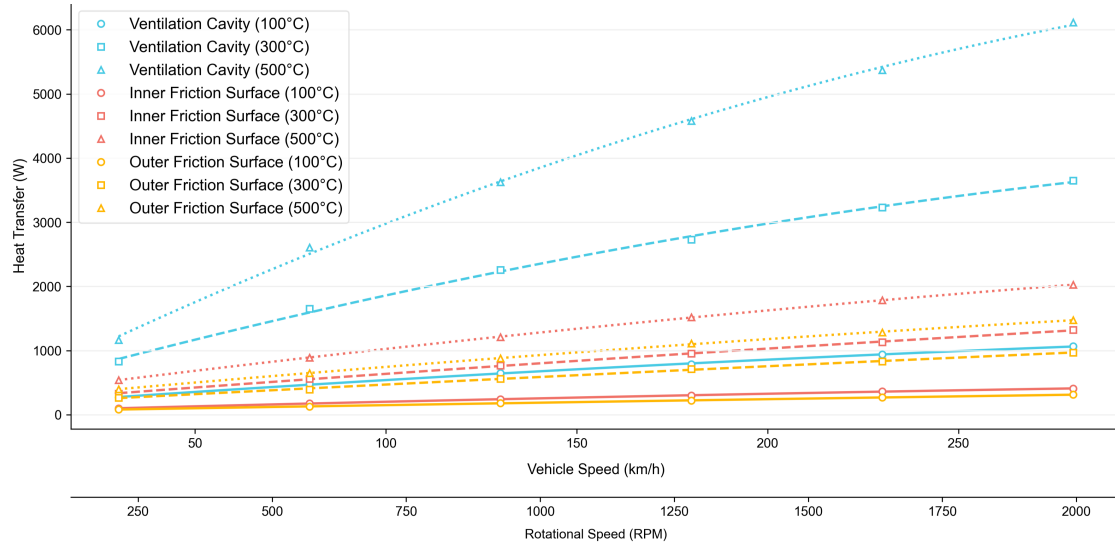


Figure B.1: Surface averaged heat transfer as a function of rotational speed and temperature ($300 \text{ m}^3/\text{h}$)

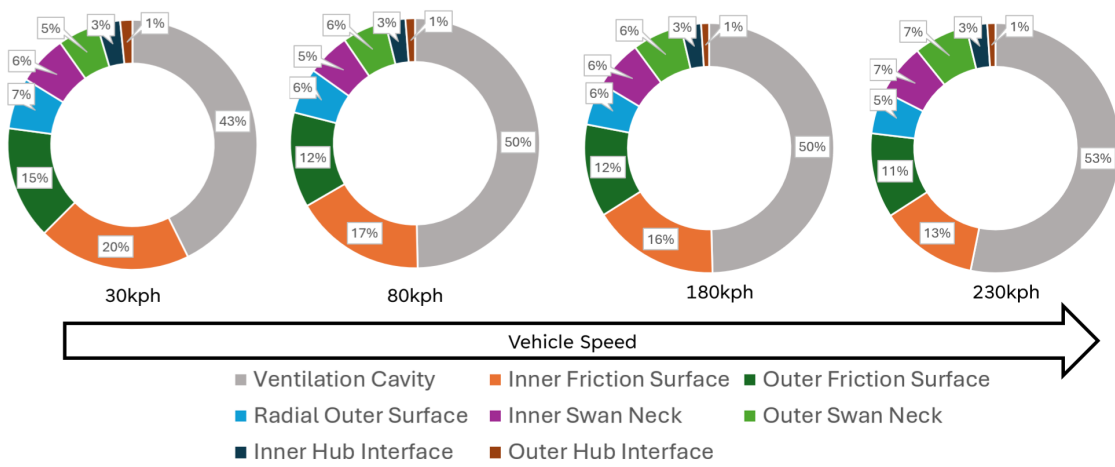


Figure B.2: Heat transfer distribution across the speed spectrum (500 °C, $300 \text{ m}^3/\text{h}$)

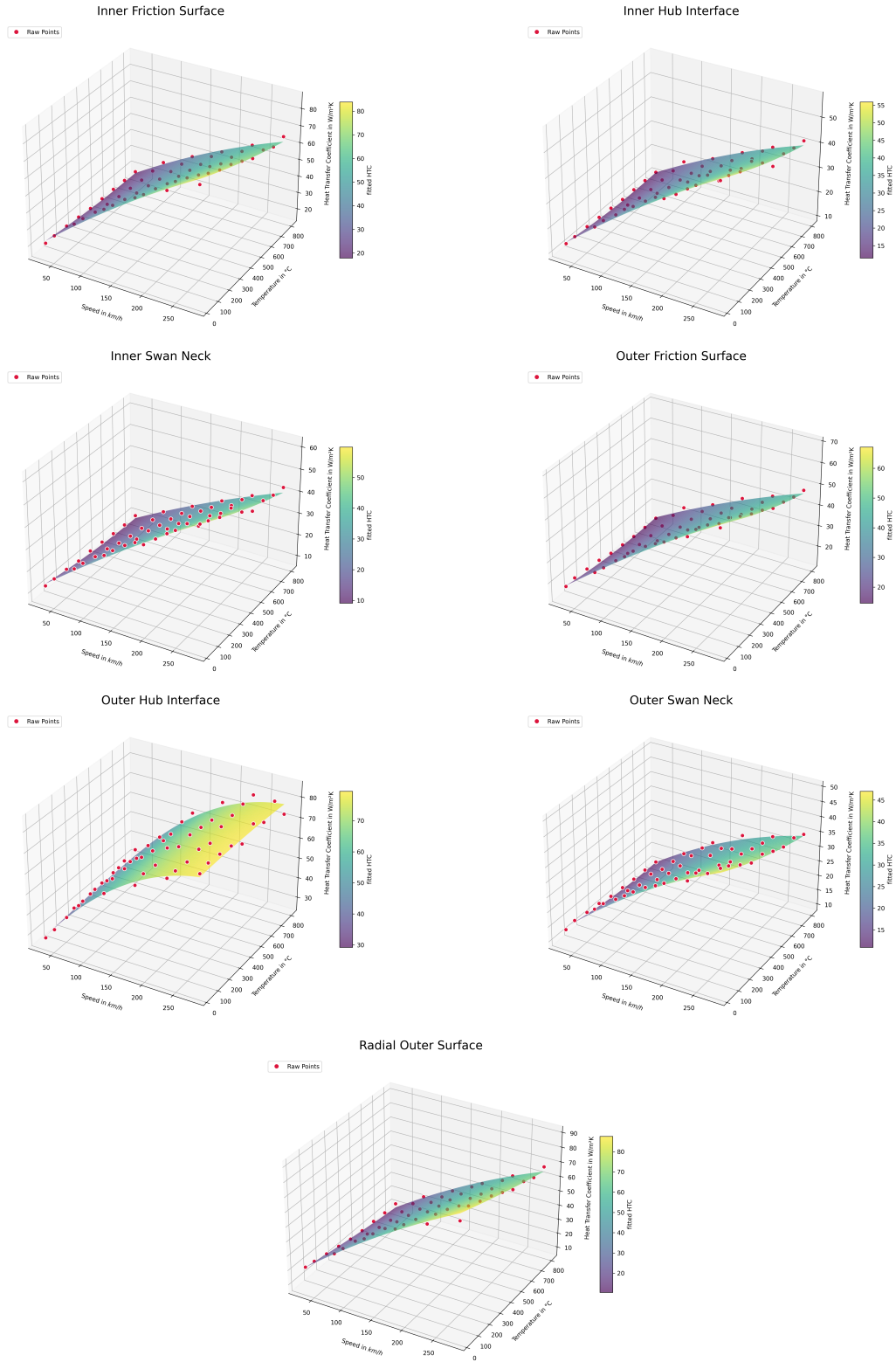


Figure B.3: Surface response plots for segmented surface using the generalized equation 6.1

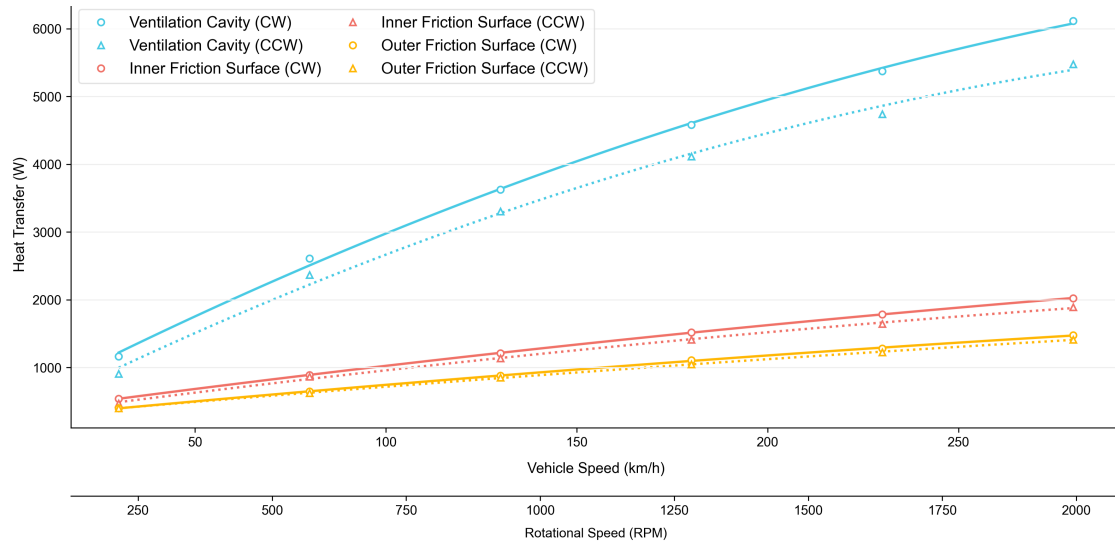


Figure B.4: Surface averaged heat transfer as a function of rotational speed and direction (500 °C, 300 m³/h)

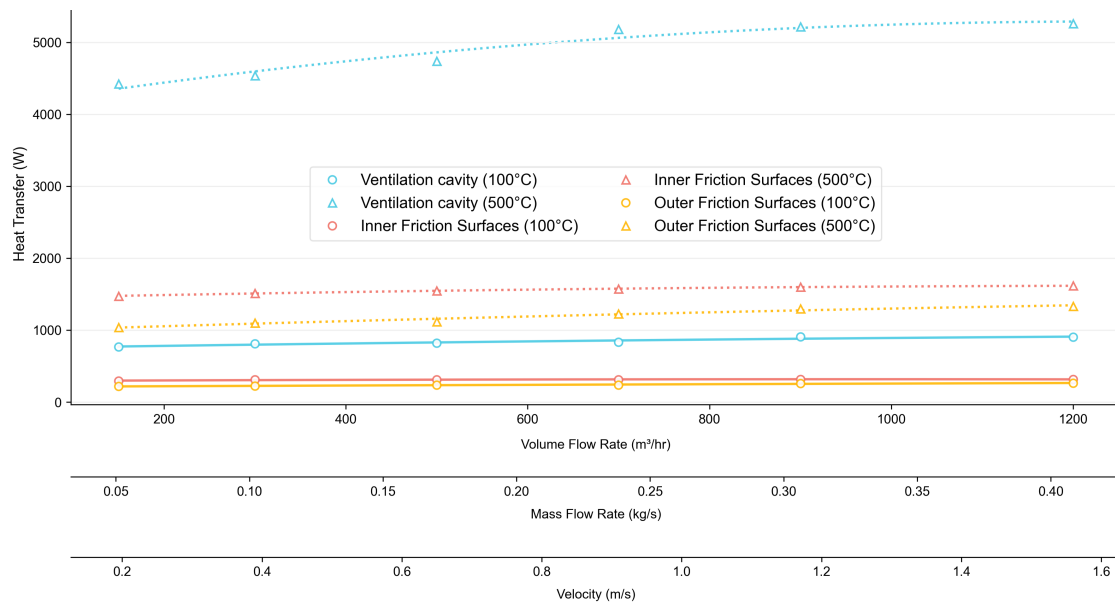


Figure B.5: Surface averaged heat transfer as a function of test-bench inlet airflow rate and temperature (180 km/h)

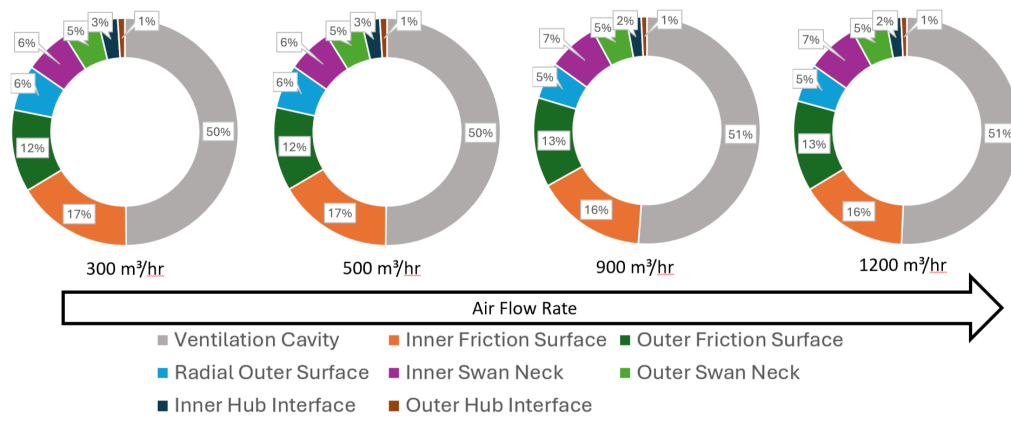


Figure B.6: Heat transfer distribution across the test-bench inlet airflow rate spectrum (500 °C, 180 km/h)

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2026

www.chalmers.se



CHALMERS
UNIVERSITY OF TECHNOLOGY