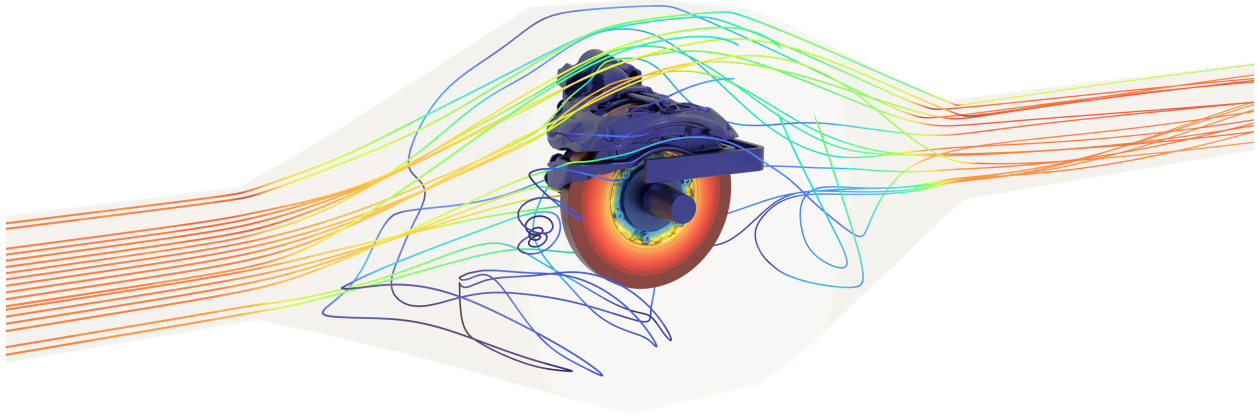




CHALMERS
UNIVERSITY OF TECHNOLOGY



Transient Conjugate Heat Transfer CFD Simulations of an Experimental Brake Rig

Focus on Modeling of Radiation and Rotation

Master's Thesis in Applied Mechanics

Linus Börjesson
Fredrik Hansson

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

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

MASTER'S THESIS 2026

Transient Conjugate Heat Transfer CFD Simulations of an Experimental Brake Rig

Focus on Modeling of Radiation and Rotation

LINUS BÖRJESSON
FREDRIK HANSSON



CHALMERS
UNIVERSITY OF TECHNOLOGY

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

Transient Conjugate Heat Transfer CFD Simulations of an Experimental Brake Rig
Focus on Modeling of Radiation and Rotation
LINUS BÖRJESSON
FREDRIK HANSSON

© LINUS BÖRJESSON, FREDRIK HANSSON, 2026.

Supervisors: Martin Petersson, Volvo Group AB
Alexey Vdovin, Department of Mechanics and Maritime Sciences

Examiner: Alexey Vdovin, Department of Mechanics and Maritime Sciences

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

Cover: Visualization of a simulated flow and surface temperatures of a brake system
in an experimental rig modeled in *Siemens Star-CCM+*.

Typeset in L^AT_EX
Printed by Chalmers Reproservice
Gothenburg, Sweden 2026

Transient Conjugate Heat Transfer CFD Simulations of an Experimental Brake Rig Focus on Modeling of Radiation and Rotation

LINUS BÖRJESSON

FREDRIK HANSSON

Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Abstract

This master's thesis investigates transient conjugate heat transfer CFD modeling of an experimental truck brake rig, with focus on how thermal radiation and brake rotation should be modeled for accurate and efficient temperature prediction. A numerical model of the rig was developed in *Simcenter STAR-CCM+* and evaluated using two braking cycles: a short severe cycle and a longer cycle adapted from the Alpine descent test.

Two approaches for modeling disc rotation were compared: Rigid Body Motion (RBM) and Convective Velocity Option (CVO). In addition, different implementations of surface-to-surface radiation were assessed in terms of temperature prediction and computational cost. The results show that radiation has a comparatively small influence during short braking events, but becomes more important in longer high-temperature cycles, where neglecting radiation leads to over-predicted brake temperatures. For RBM, radiation had to be modeled in the solid domain to ensure energy conservation, although this significantly increased computational cost. To mitigate this, a parameter sweep was conducted to optimize the radiation settings. By reducing the *Patch/Face Proportion* and the *Number of Beams*, the solver computational time was decreased by more than 14.8%. This optimization maintained a high level of accuracy, yielding a Root Mean Square Error below 0.5 for the predicted temperatures compared to the most computationally expensive baseline.

The results also show that CVO predicts lower maximum temperatures than RBM due to numerical smearing, while average disc temperatures remain relatively close. However, CVO reduced computational time by about 84 to 86%, making it a worthy engineering compromise when simulation speed is important. Comparison with brake rig measurements showed that both approaches captured the overall temperature development during braking, but both under-predicted temperatures during cooling. The thesis concludes that radiation should be included in transient brake thermal simulations, especially for long-duration braking, and that CVO is a promising alternative to RBM while, balancing accuracy and efficiency.

Acknowledgements

We would like to thank everyone who has contributed to this Master's thesis, which was carried out as a collaboration between Chalmers University of Technology and Volvo Group.

First, we would like to thank our examiner and supervisor at Chalmers, Alexey Vdovin, for sharing his expertise in CFD and for guiding us throughout this project. We would also like to thank Nandu Suresh and Jonas Sjöblom for their work on the experimental rig and for providing the verification data. We wish them the best in the future development of the rig.

At Volvo Group, Sudharsan Vasudevan has been invaluable in helping us set up our models and in providing feedback on every part of this thesis, for which we are very grateful. We would also like to thank Martin Peterson at Volvo Group for his general support and expertise in disc brakes. In addition, we would like to thank Torbjörn Wiklund and Sassan Etemad at Volvo Group for sharing their expertise and for the valuable discussions on CFD and heat transfer analysis.

We would also like to thank our fellow Master's thesis students at Volvo Group, Melvin Glans and Emil Sjöström, for their valuable feedback on our work. Finally, we would like to express our sincere gratitude to Volvo Group and to our group manager, Ingela Stjärnväy, for giving us the opportunity to carry out this project.

Linus Börjesson, Fredrik Hansson, Gothenburg, June 2026

List of Acronyms

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

CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
CPU	Central Processing Unit
CVO	Convective Velocity Option
DNS	Direct Numerical Simulation
DPF	Diesel Particulate Filter
ELPI	Electrical Low Pressure Impactor
HPC	High-Performance Computing
HTC	Heat Transfer Coefficient
LES	Large Eddy Simulation
PID	Proportional-Integral-Derivative
PM	Particulate Matter
PM _{2.5}	Particulate Matter smaller than $2.5\mu m$
PM ₁₀	Particulate Matter smaller than $10\mu m$
RANS	Reynolds-Averaged Navier-Stokes
RBM	Rigid Body Motion
RMSE	Root Mean Square Error
TVS	Tangential Velocity Specification

Nomenclature

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

List of Symbols

A	Area [m ²]
α_{slope}	Slope Angle [°]
C_d	Drag Coefficient [-]
C_p	Specific Heat [J/kgK]
C_{rr}	Rolling Resistance Coefficient [-]
$d\mathbf{a}$	Outward Pointing Area Vector [m ²]
E	Energy Density [J/m ³]
e	Thermal Effusivity [J/m ² Ks ^{1/2}]
ϵ	Dissipation Rate [W/kg]
F_{aero}	Aerodynamic Drag Force [N]
F_{brake}	Brake Force [N]
F_g	Gravitational Force [N]
F_{other}	Miscellaneous Force [N]
F_{resist}	Resistance Force [N]
F_{roll}	Rolling Resistance [N]
F_{rad}	Radiation Factor [-]
g	Gravitational Acceleration [m/s ²]
h	Standard State Enthalpy [J/kg]
H	Heat Transfer Coefficient [W/m ² K]
i	Summation Index [-]
$\mathbf{I}$	Identity Tensor [-]
J	Diffusional Flux [mol/m ² s]

j	Index for Dimension [-]
κ	Thermal Conductivity [W/mK]
k	Turbulent Kinetic Energy [J/kg]
m	Mass [kg]
M	Torque [Nm]
M_{molar}	Molar Mass [kg/mol]
n	Species Index [-]
n_{brakes}	Number of Brakes [-]
N	Normal Force [N]
ω	Angular Velocity [rad/s]
p	Pressure [Pa]
P	Power [W]
ϕ	Scalar [-]
Q	Heat Rate [W]
$\mathbf{q}''$	Heat Transfer [W/m ²]
ρ	Density [kg/m ³]
r_{tyre}	Tyre Radius [m]
s	Volumetric Heat Generation [W/m ³]
S_h	Source Term [J/m ³ s]
σ	Stefan Boltzmann Constant [W/m ² K ⁴]
t	Time [s]
T	Temperature [K]
T^+	Non-dimensional Temperature [-]
τ	Viscous Stress [Pa]
τ_w	Wall Shear Stress [Pa]
θ	Rotation Angle [rad]
u	Velocity x-direction [m/s]
u_*	Friction Velocity [m/s]
U	Velocity Field [m/s]
v	Velocity y-direction [m/s]
V	Volume [m ³]
$\mathbf{v}_s$	Convective Velocity [m/s]
ν	Kinematic Viscosity [m ² /s]
w	Velocity z-direction [m/s]

x	Spatial Coordinate [m]
y	Wall Distance [m]
y_c	Distance to Near Wall Center [m]
y^+	Dimensionless Wall Distance [-]

Contents

List of Acronyms	ix
Nomenclature	xi
List of Figures	xix
List of Tables	xxi
1 Introduction	1
1.1 Background	1
1.2 Aim	2
1.3 Limitations	2
1.4 Thesis Questions	2
2 Theory	3
2.1 Brake System	3
2.1.1 Braking Parameters	4
2.2 Governing Equations	5
2.2.1 Momentum Equations	5
2.2.2 Energy Equation	5
2.2.3 Continuity Equation	5
2.3 Turbulent Flow	6
2.3.1 Direct Numerical Simulation	7
2.3.2 Large Eddy Simulation	7
2.3.3 Reynolds-Averaged Navier-Stokes	7
2.3.3.1 Realizable K-Epsilon Two-Layer Turbulence	8
2.4 Modeling of Rotation	8
2.4.1 Rigid Body Motion	9
2.4.2 Convective Velocity Option	9
2.4.3 Tangential Velocity Specification	9
2.5 Heat Transfer	9
2.5.1 Conduction	10
2.5.2 Convection	10
2.5.2.1 Heat Transfer Coefficients	11
2.5.3 Thermal Radiation	11
2.6 Thermal Effusivity	11
2.7 Discretization	12

2.7.1	Surface Wrapper	12
2.7.2	Prism Layer	12
2.7.3	Thin Mesher	12
2.7.4	Surface Remesher	13
2.7.5	Automatic Surface Repair	13
2.7.6	Polyhedral Mesher	13
2.8	Conjugate Heat Transfer	13
2.8.1	Energy Coupling	14
2.9	Computational Scalability	14
3	Methodology	17
3.1	Braking Cycles	17
3.1.1	Disc Crack Test	17
3.1.2	Alpine Descent	17
3.1.2.1	The 900/1800 Braking Cycle	18
3.1.2.2	Rig Braking Cycle	18
3.2	Experimental Setup	18
3.2.1	Geometry	18
3.2.2	Measurements	19
3.2.3	Operating Limits	20
3.3	Numerical Setup	20
3.3.1	Geometry	20
3.3.2	Mesh	23
3.3.2.1	Surface Mesh	24
3.3.2.2	Volume Mesh	24
3.3.2.3	y^+	25
3.3.3	Physics	26
3.3.3.1	Flow Solver	27
3.3.3.2	Turbulence	27
3.3.3.3	Rotation	27
3.3.3.4	Regions	27
3.3.3.5	Radiation	28
3.3.3.6	Energy Coupling	29
3.3.3.7	Boundaries	29
3.3.3.8	Heat Application	30
3.3.3.9	Materials	30
3.3.4	Monitored Variables	31
3.3.4.1	Thermal Probes	31
3.3.4.2	Maximum and Average Temperatures	31
3.3.4.3	Heat Rates	32
3.3.4.4	Computational Time	32
3.3.4.5	Scenes	32
3.3.5	Simulation	32
3.3.5.1	Time Scales	32
3.3.5.2	Stages	33
3.3.5.3	Solvers	33

3.3.5.4	Simulation Operation Base Setup	33
3.3.5.5	Scalability of Computational Power	34
3.4	Investigations	34
3.4.1	Effects of Implementing Radiation	34
3.4.2	Optimized Radiation	35
3.4.3	Convective Velocity Option	36
3.4.4	Verification of Simulations	36
4	Results	39
4.1	Effects of Implementing Radiation	39
4.1.1	Setup A	39
4.1.2	Setup B	41
4.1.2.1	40/60	41
4.1.2.2	900/1800	43
4.2	Optimized Radiation	44
4.2.1	Simulation 1-4	44
4.2.2	Simulations 5-6	46
4.2.3	Visualization	47
4.2.3.1	40/60	47
4.2.3.2	900/1800	49
4.3	Convective Velocity Option	51
4.3.1	40/60	51
4.3.2	900/1800	53
4.3.3	Heat Application	55
4.3.4	Temperature	56
4.4	Verification of Simulations	57
5	Discussions	61
5.1	Temperature Monitors	61
5.2	Radiation	61
5.2.1	Radiation Modeling Approach	62
5.2.1.1	Time Savings	62
5.2.2	Thermal Effects of Radiation	62
5.2.2.1	Time Savings	63
5.2.3	Optimized Radiation	63
5.2.3.1	Parameters	64
5.2.3.2	Time Savings	64
5.3	Heat Transfer	65
5.3.1	Choice of Energy Coupling	65
5.3.2	Fluctuating Convection	65
5.3.3	Oscillating Convection	66
5.3.4	Heat Application	66
5.4	Convective Velocity Option	67
5.4.1	Temperature	67
5.4.2	Heat Rates	67
5.4.3	Difference in Flow	68
5.4.4	Heat Application	68

5.4.5	Time Savings	69
5.5	Experimental Rig	69
5.5.1	Temperature Measurements	69
5.5.2	Geometry and Materials	70
6	Conclusions	71
6.1	Radiation	71
6.2	Energy Coupling	71
6.3	Flow Solver	72
6.4	Convective Velocity Option	72
6.5	Verification with Rig	72
6.6	Future Work	73
	Bibliography	73

List of Figures

2.1	Schematic of a truck brake system	4
2.2	Illustration of a conjugate heat transfer coupling approach	14
3.1	Geometry of the experimental brake rig	19
3.2	Locations of temperature sensors in the disc	20
3.3	Geometry of enclosure and the brake system in the numerical setup .	21
3.4	Brake system in numerical setup - left view	22
3.5	Brake system in numerical setup - top view	22
3.6	Surfaces of the wrapped air domain	23
3.7	Surface mesh of the air domain	24
3.8	Surface mesh of the solid domain without the enclosure	24
3.9	Cut of volume mesh of the air domain with zoomed in figure, showing the prism layers	25
3.10	Cut of volume mesh of the solid domain - left view	25
3.11	y^+ scene of the numerical setup	26
3.12	Solid regions for the RBM setup	28
3.13	Solid regions for the CVO setup	28
3.14	The interface between brake disc and brake pad where the heat is applied	30
3.15	Thermal probes placements in the simulations	31
4.1	Maximum and average temperature in the disc with and without radiation for Setup A, simulating the 40/60 braking cycle	39
4.2	The heat rates leaving the rotating parts for Setup A, simulating the 40/60 braking cycle	40
4.3	The total solver CPU time for Setup A with and without radiation, simulating the 40/60 braking cycle	40
4.4	Heat rates leaving the rotating parts with static cooling for Setup A, simulating the 40/60 braking cycle	41
4.5	Maximum and average temperature in the disc with and without radiation for Setup B, simulating the 40/60 braking cycle	41
4.6	Heat rates leaving the rotating parts for simulations with and without radiation for Setup B, simulating the 40/60 braking cycle	42
4.7	Total CPU time for simulations with and without radiation for Setup B, simulating the 40/60 braking cycle	42
4.8	Maximum and average temperature in the disc with and without radiation for Setup B, simulating the 900/1800 braking cycle	43

4.9	Heat rates leaving the rotating parts for simulations with and without radiation for Setup B, simulating the 900/1800 braking cycle	43
4.10	Total solver CPU time for simulations with and without radiation for Setup B, simulating the 900/1800 braking cycle	44
4.11	Maximum and average temperature in the disc with different radiation parameters for Setup B, simulating the 900/1800 braking cycle .	45
4.12	Total solver CPU time for simulations with different radiation parameters for Setup B, simulating the 900/1800 braking cycle	45
4.13	Total, convective and radiative heat rates in the rotating parts with different radiation parameters for Setup B, simulating the 900/1800 braking cycle	46
4.14	Total solver CPU time for all set of radiation settings for Setup B, simulating the 900/1800 braking cycle	47
4.15	Radiative heat flux for 40/60 braking cycle at 40 seconds	48
4.16	Temperature of the brake system for the 40/60 braking cycle at 30, 40 and 50 seconds	48
4.17	Air flow for the 40/60 braking cycle at 100 seconds - front view . . .	49
4.18	Air flow for the 40/60 braking cycle at 100 seconds - top view	49
4.19	Surface temperature development over time from 300 to 1200 seconds	50
4.20	Air flow evolution over time during cooling stage of the 900/1800 braking cycle in the RBM setup	50
4.21	Maximum and average temperature in the disc with RBM and CVO for the 40/60 braking cycle	51
4.22	Total solver CPU time with RBM and CVO for the 40/60 braking cycle	51
4.23	Total, convective and radiative heat rates from the rotating parts for the RBM and CVO setups for 40/60 braking cycle	53
4.24	Maximum and average temperature in the disc with RBM and CVO for the 900/1800 braking cycle	53
4.25	Total solver CPU time plot with RBM and CVO for the 900/1800 braking cycle	54
4.26	Total, convective and radiative heat rates from the rotating parts for the RBM and CVO setups for the 900/1800 braking cycle	55
4.27	Heat generated for the RBM setup at the disc surface	56
4.28	Heat generated for the CVO setup at the disc surface	56
4.29	Surface temperature comparison between the RBM (left figure) and CVO (right figure) setups at 900 seconds	57
4.30	Surface temperature comparison between the RBM (left figure) and CVO (right figure) setups at 1200 seconds	57
4.31	Comparison of thermal probes in the rig, T1-T5, and the point probes in the simulations	58
4.32	All measured individual temperatures measured in the brake rig, T1-T5	59
5.1	Comparison between rig and simulations with just T1/T2 and T3/T4	70

List of Tables

3.1	Mesh default controls	23
3.2	Material properties of the used materials	30
3.3	Simulated setups for investigating the effects of implementing radiation	35
3.4	Parameter values of the manual parameter sweep	36
3.5	Parameter values of the extra simulations	36
4.1	Thermal energy applied to the brake system and energy leaving the rotating parts for Setup B, simulating the 40/60 braking cycle	42
4.2	Thermal energy applied to brake system and energy leaving the ro- tating parts for Setup B, simulating the 900/1800 braking cycle . . .	44
4.3	Comparison of the different radiation parameter settings	45
4.4	The RMSE and time savings of Simulations 5 and 6 for Setup B, simulating the 900/1800 braking cycle	47
4.5	The RMSE and time savings for the CVO setups compared to the RBM setup for the 40/60 braking cycle	52
4.6	The RMSE and time savings for the CVO setups compared to the RBM setup for the 900/1800 braking cycle	54
4.7	The temperature after braking and cooling for the simulations com- pared to the rig	58

1

Introduction

This chapter introduces the thesis topic, its relevance, and its scope. Followed by a presentation of the limitations and thesis questions.

1.1 Background

The upcoming Euro 7 regulations introduce new challenges for truck manufacturers. These regulations set limits on particle emissions due to brake wear and tire abrasion for motor vehicles [1]. For Volvo Group, this creates the need for predictive tools that can support the analysis and reduction of particulate emissions from brakes [2].

Particles are classified by size where the most important categories are Particulate Matter (PM) smaller than $10\text{ }\mu\text{m}$ (PM_{10}) and PM smaller than $2.5\text{ }\mu\text{m}$ ($\text{PM}_{2.5}$). Therefore, $\text{PM}_{2.5}$ is a subcategory of PM_{10} . These particles are often airborne and may pose significant health risks when inhaled. Deposition of these particulates and where they become trapped in the respiratory system depends directly on their size [3]. Around 50% of the particulate emissions from brake wear become airborne and among these, around 80% are of size PM_{10} . A mass-weighted mean of the particle size distribution is between 2 and $6\text{ }\mu\text{m}$ [4].

Particulate emissions from brakes are strongly influenced by the temperature of the disc and pads. As the brake disc and pad temperature increases from 25°C to 100°C , brake wear increases by 2-10 times [5]. It is therefore important to understand and predict the thermal behavior of the brake system in order to support efforts to reduce particulate emissions from brakes. Common cooling strategies include ventilated discs, air ducts, and rim designs that redirect air towards the brakes. The hot surfaces in the braking system are cooled by forced convection heat transfer with the surrounding air. Increasing the air flow around these parts results in increased Heat Transfer Coefficient (HTC), which in turn enhances cooling.

In recent years, the demand for electric trucks has increased, resulting in new challenges for the automotive industry. Through regenerative braking, electric vehicles can recover some of the braking energy as electrical energy and reduce emissions from brake wear. The regenerated electrical energy is stored in the battery, which extends the driving range [6]. However, if the batteries are fully charged, regenerative braking cannot be used since the electrical energy cannot be stored [7]. As a result, friction braking remains important in certain operating conditions, particularly in

demanding scenarios where regenerative braking is limited or unavailable. Therefore, understanding and analyzing brake thermal behavior and particulate emissions remain important, even for electric trucks.

To support the work in reducing particulate emissions from brakes, Energimyndigheten has initiated a project in collaboration with Volvo Group, Haldex AB, and Chalmers University of Technology. This project consists of 5 work packages on both fluid and solid analyses and involves one PhD student as well as several related collaborations. The first step in the fluid-dynamics work is to establish an accurate model that predicts the temperature of the brake system in a rig. The rig is located at Chalmers and is used to simulate different braking cycles in order to measure both temperatures and particulate emissions from brakes. The collaboration aims to improve understanding of particle emissions in order to reduce them [8].

1.2 Aim

This thesis aims to predict temperatures in a truck brake system used in an experimental brake rig using Computational Fluid Dynamics (CFD) and Conjugate Heat Transfer (CHT) simulations in *Star-CCM+*. Focus is placed on how thermal radiation and brake rotation should be modeled to enable accurate simulations. The results are intended to support future work on the prediction of particulate emissions from brakes.

1.3 Limitations

The thesis is limited primarily by the 20-week project duration and therefore focuses only on modeling the experimental brake rig at Chalmers University of Technology using a solid brake disc and not a ventilated disc. It is further limited to thermal analysis of the brake system and does not include direct analysis of particle emissions. Furthermore, the model will not cover any structural analysis and will therefore treat the solids as rigid bodies. In other words, no elastic or plastic deformation due to temperature or loading will be modeled.

1.4 Thesis Questions

The thesis is intended to address the following research questions:

- How is radiation modeled in a transient brake rig CFD-CHT model?
- How does radiation influence predicted brake temperatures and computational cost?
- Can Convective Velocity Option (CVO) provide an acceptable alternative to Rigid Body Motion (RBM) for this application?
- To what extent can the simulation predict brake temperatures measured in the experimental rig?

2

Theory

This chapter presents the relevant theory to accurately simulate a brake system. It covers how brake systems work, governing equations for flow, turbulence modeling, rotation modeling, heat transfer and discretization. Furthermore, this chapter also covers how a CHT analysis is formulated and coupled between a solid and a fluid domain, and the scalability of computational power.

2.1 Brake System

The function of brakes is to decelerate or stop a vehicle by converting kinetic and potential energy into heat through friction between the brake disc and brake pads [9]. Volvo uses a pneumatic disc brake system for heavy-duty trucks, consisting of the following main parts, shown in Figure 2.1 [10].

1. **Brake disc:** The disc follows the rotation of the wheel. This is where braking takes place.
2. **Brake pads:** Mounted in the caliper, they clamp the disc when braking, resulting in friction.
3. **Brake caliper:** Main body of the system. It can either be fixed or floating.
4. **Hub:** The connection between the wheels and the disc.
5. **Caliper piston:** Receives mechanical force from the actuator and pushes the pads toward the disc.
6. **Actuator/Chamber:** Converts air pressure into mechanical force.
7. **Air system:** Consists of compressors, air tanks, regulators, brake valves and air lines. Delivers pressurized air to the actuator.

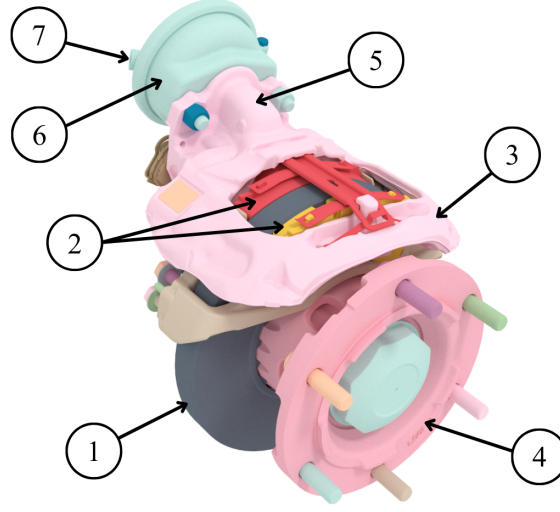


Figure 2.1: Schematic of a truck brake system

2.1.1 Braking Parameters

To calculate the braking torque required to be applied for a vehicle traveling at constant velocity on a slope with a given angle, first, the relevant forces need to be identified to find the brake force, F_{brake} . These forces could be divided into the gravitational force, F_g , that drives the vehicle downhill and a resistance force, F_{resist} .

$$F_{brake} = F_g - F_{resist} \quad (2.1)$$

On a specific slope, the gravitational force can be computed as [11]:

$$F_g = mg \sin(\alpha_{slope}) \quad (2.2)$$

The resistance force can be further divided into different components. One way of doing this is to divide it into a drag force, a roll resistance force and a term for other miscellaneous forces presented in the equation below [11].

$$F_{resist} = F_{aero} + F_{roll} + F_{other} \quad (2.3)$$

The drag force and the roll resistance force can be calculated as the equation below [11].

$$F_{aero} = \frac{1}{2} \rho A C_d U^2 \quad (2.4)$$

$$F_{roll} = C_{rr} N = C_{rr} mg \cos(\alpha_{slope}) \quad (2.5)$$

From the braking force, the torque for each wheel can be calculated as:

$$M = \frac{F_{brake} r_{tyre}}{n_{brakes}} \quad (2.6)$$

Given the brake torque, the heat generated between the pads and the disc can be approximated as the following equation [12].

$$P = M \frac{d\theta}{dt} = M\omega \quad (2.7)$$

2.2 Governing Equations

For a compressible Newtonian fluid, a CFD simulation uses three different governing equations to solve the flow. These are conservation of momentum, energy, and continuity equations. When the rate of accumulation and convective transport is balanced by diffusive transport and source terms, the governing equations are satisfied. For the three equations the index j represents the dimensions and is equal to 1, 2, 3 for a three-dimensional case [13, p. 8].

2.2.1 Momentum Equations

The momentum equation describes the balance of momentum in a fluid control volume. It shows that the rate of accumulated momentum is equal to the rate of momentum entering the element, the momentum leaving the element, and the sum of forces acting on the element. For a Newtonian fluid, the governing equation for conservation of momentum is known as the Navier-Stokes equation and it reads as follows [13].

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (2.8)$$

The left-hand side describes acceleration where the first term represents the change of velocity over time and the second term represents the convective acceleration. The first term on the right-hand side is the pressure-gradient term, which represents forces due to pressure differences. The second term is the diffusion term that represents the diffusion of momentum due to viscous forces [13, pp. 14-16].

2.2.2 Energy Equation

The energy equation describes energy conservation in a control volume for a transient system. The total energy E can be described as the sum of the kinetic, thermal, chemical, and potential energies. In balanced form, the governing equation for conservation of energy reads as follows [13, p. 16].

$$\frac{\partial E}{\partial t} = -\frac{\partial}{\partial x_j} \left[EU_j - \kappa \frac{\partial T}{\partial x_i} + \sum_n M_{molar_n} h_n J_n - \tau_{kj} U_k \right] + S_h \quad (2.9)$$

The left-hand side represents the change of total energy over time. The terms on the right-hand side are the convective transport term, conductive heat flux term, change of standard state enthalpy, viscous work term and source term [13, p. 16].

2.2.3 Continuity Equation

The continuity equation represents conservation of mass in a control volume. The physical interpretation is that the accumulated mass is equal to the mass entering and leaving the fluid element, and the equation is written as follows [13, pp. 12-14].

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_j}{\partial x_j} = 0 \quad (2.10)$$

2.3 Turbulent Flow

Turbulence is a common flow phenomenon in engineering applications. It can be characterized by three-dimensional structures, occurring at large Reynolds numbers, irregularity, dissipation of turbulent kinetic energy and diffusivity.[13, pp. 63-66].

Irregularity refers to the fact that turbulent flows are irregular, chaotic and random. In the flow, a wide range of time scales, velocity scales and length scales occur. The different length scales are often referred to as turbulent eddies or vortices and can be described as a motion that is to some extent coherent over a certain region. Different length scales coexist in the flow which means that smaller eddies can occur inside larger eddies. The smaller eddies are bounded by viscous effects while the larger eddies, which may be several orders of magnitude larger, are bounded by the geometry of the flow. Because of the different length scales and irregularity of the flow it is difficult to model the flow with a deterministic approach [13, pp. 63-64].

Dissipation of turbulent kinetic energy occurs in all turbulent flows. Energy is extracted from the mean flow and enters the turbulence in the largest eddies. The energy is then transferred by inviscid processes into smaller and smaller eddies because the larger eddies are unstable and break down to smaller ones. By viscous action due to molecular viscosity, the energy is dissipated into viscous heat at the smallest scales [13, pp. 65-66].

One of the most important characteristics of turbulence is enhanced diffusion. Due to the chaotic fluid motion, turbulent transport can produce mixing rates for species, momentum, and energy that are several orders of magnitude higher than those associated with molecular diffusion [13, p. 64].

To model a turbulent flow, special attention must be given to the regions close to walls where a turbulent boundary layer is created. In this layer the velocity increases rapidly from zero to the free stream velocity. The boundary layer can then be divided into an inner and an outer region. The inner region closest to the wall is then divided into sub-layers based on the wall friction velocity u_* and the dimensionless wall distance y^+ . These quantities can be calculated as a function of the wall shear stress τ_w , density ρ , kinematic viscosity ν and wall distance y shown below [13, pp. 101-103].

$$u_* = \sqrt{\frac{\tau_w}{\rho}} \quad (2.11)$$

$$y^+ = \frac{u_* y}{\nu} \quad (2.12)$$

There are three different sub-layers in the inner region. The closest to the wall is the viscous sub-layer with $0 < y^+ < 5$. This is followed by the buffer sub-layer with $5 < y^+ < 30$. Finally, the log-law sub-layer with $30 < y^+ < 400$. When discretizing the domain, it is possible to select the extent to which the flow in the sub-layer should be resolved by choosing the first cell height to a desired y^+ [13, p. 103].

2.3.1 Direct Numerical Simulation

Simulating turbulent flow using Direct Numerical Simulation (DNS) seems the obvious choice of method at first glance. In this method, the unsteady Navier-Stokes equations are solved directly and therefore no turbulence model is required. Since turbulent flows contain a wide range of length and time scales, very short time-steps and dense computational grids would be required to resolve the flow properly. In combination with non-linear equations, a large amount of computational power is needed to solve the problem. In fact, it would not be possible to perform DNS for a real engineering problem with the computers used today [13, p. 79].

2.3.2 Large Eddy Simulation

Since the challenges of DNS is that very short time-steps and dense computational grids are required to resolve the smallest length and time scales in the flow, Large Eddy Simulation (LES) can be used instead. When using LES, the largest and intermediate turbulent scales are resolved, while the smaller scales are modeled. Approximately 80% of total turbulence energy is resolved. The smallest turbulence scales remain unresolved, but can be modeled using universal subgrid models. As a result, the larger turbulence scales that are more complicated to model, can then be simulated directly. The LES method requires less computational power compared to DNS. However, a fine grid with short time-steps is still needed to obtain good results. As a consequence, the computational cost is still high compared to other turbulence models [13, pp. 79-80].

2.3.3 Reynolds-Averaged Navier-Stokes

In the Reynolds-Averaged Navier-Stokes (RANS) equations, each solution variable ϕ is decomposed into a mean part and a fluctuating part. The solution variable can be pressure, velocity components, energy or species concentration. The decomposition is represented in the equation below [14].

$$\phi = \bar{\phi} + \phi' \quad (2.13)$$

For steady-state situations, the averaging can be seen as time-averaging and for transient situations, it can be seen as taking the average of many independent realizations. For the mean quantities, the averaged equations are identical in form to the original equations except that in the momentum equation, an extra term appears. It is the Reynolds stress tensor and is defined in the equation below [14].

$$\mathbf{T}_{RANS} = -\rho \begin{bmatrix} \overline{u'u'} & \overline{u'v'} & \overline{u'w'} \\ \overline{v'u'} & \overline{v'v'} & \overline{v'w'} \\ \overline{w'u'} & \overline{w'v'} & \overline{w'w'} \end{bmatrix} + \frac{2}{3}\rho k\mathbf{I} \quad (2.14)$$

The main challenge is to model this stress tensor, which offers closure for the governing equations [14].

2.3.3.1 Realizable K-Epsilon Two-Layer Turbulence

K-Epsilon turbulence models introduce transport equations for the turbulent kinetic energy k and dissipation rate ϵ . These equations are then solved to determine the turbulent eddy viscosity. Several variants of the K-Epsilon model exist, one of which is the *Realizable K-Epsilon Two-Layer* model [14].

K-Epsilon models can use either a high or low Reynolds number approach. In the high Reynolds number approach the K-Epsilon model is applied with wall functions. However, this method does not account for the blocking effects of the viscous and buffer layers. Hence, the low Reynolds number approach was introduced where damping functions are applied to the coefficients in the model. As a function of the Reynolds number and often wall distance, the damping functions tune the coefficients in the model. An alternative to the low Reynolds number approach is the *K-Epsilon Two Layer* turbulence model, that enables application of the K-Epsilon model in both the viscous sub-layer and buffer layer [14].

As the name suggests, the two layer model divides the computation into two layers. There is the layer next to the wall where the turbulent viscosity μ_t and dissipation rate ϵ are calculated as a function of the wall distance. The transport equations are solved in the region far from the wall, and these values are then mixed with the calculated turbulent dissipation rate close to the wall. This allows the flow properties to be solved in the entire domain. The two-layer approach can be used in *Star-CCM+* with both low Reynolds number meshes and wall function type meshes, with $y^+ \approx 1$ and $y^+ > 30$, respectively [14].

A modified version of the K-Epsilon model, called *Realizable K-Epsilon*, ensures that normal stresses remain within physically allowable limits. It introduces a new transport equation for the turbulent dissipation rate and a variable damping function f_u . The damping function is applied to the model coefficients and is a function of the mean flow and turbulence properties [14].

The *Realizable K-Epsilon Two-Layer* model combines the *Realizable K-Epsilon* and *Two-Layer K-Epsilon* formulations. This model is known to perform well in free-stream regions, although it may exhibit reduced accuracy in predicting near-wall effects and flow separation [15].

2.4 Modeling of Rotation

There are different approaches to simulating rotating objects. For the solid domain, examples include RBM and CVO while for the fluid domain, one example is Tangential Velocity Specification (TVS). The RBM approach is used for transient simulations, while TVS and CVO are methods mostly used in steady-state simulations.

2.4.1 Rigid Body Motion

The motion of an object can be modeled by moving the vertices of its mesh according to a specified rotation, translation or trajectory. For RBM, the object moves a fixed displacement per time step, thus it must be a transient analysis. The advantage of this approach is that the solution is time-accurate and captures transient effects. Thus, this is the most accurate approach for simulations involving moving parts [16]. Due to the unsteady nature of this approach and the fact that it often involves changes in the interface that needs to be recomputed, it is the most computationally intensive [17].

2.4.2 Convective Velocity Option

It is also possible to simulate the heat transfer of a moving solid without physically moving the solid. To do that, a convective term is included in the solid energy equation. The solid energy equation can then be written as follows, where $\mathbf{v}_s$ is the convective velocity [18].

$$\frac{d}{dt} \int \rho C_p T dV + \oint_A \rho C_p T \mathbf{v}_s \cdot d\mathbf{a} = - \oint_A \mathbf{q}'' \cdot d\mathbf{a} + \int_V s dV \quad (2.15)$$

In order to be able to use this method, it is important that the body subjected to the translation or rotation does not contain any holes or grooves parallel to the axis of rotation.

2.4.3 Tangential Velocity Specification

For the fluid domain, the rotation can be simulated using TVS. It is implemented through wall boundary conditions with the no-slip condition, which implies that the fluid sticks to the wall and follows its motion. To apply a rotating motion of the wall, the tangential velocity is set to the desired value. This means that motion can be applied without physically rotating the body [19]. Alternatives for modeling rotation in the fluid domain would be sliding or overset meshing techniques.

2.5 Heat Transfer

Heat transfer can be described as the study of energy transport due to temperature gradients. It is governed by the first and second law of thermodynamics. The first law states that the heat generated in a system and the net energy flow across the system boundary are equal to the change in internal energy of the system. The second law states that heat flows spontaneously from regions of higher temperature to regions of lower temperature. Heat transfer can occur through three different fundamental mechanisms simultaneously, conduction, convection, and radiation, described further below [20, pp. 1-2].

2.5.1 Conduction

Due to temperature differences in a body or between bodies without the presence of mass flow and mixing, conduction occurs. The heat rate due to conduction can be described through Fourier's law, which states that the heat rate Q in any direction is proportional to the temperature gradient and area perpendicular to the flow and is shown in the following equation [20, pp. 2-3].

$$Q = -\kappa A \frac{dT}{dx} \quad (2.16)$$

In the equation above, κ is the thermal conductivity with units of W/mK and is a measure of how much energy is transferred due to molecular interactions in the material. A good conductor has a high value of κ and therefore conducts more energy while insulators have a low value of κ and therefore conduct less energy. A is the area perpendicular to the flow in m^2 , dT is the temperature difference in kelvin and dx is the distance over which the temperature difference occurs [20, pp. 2-3].

2.5.2 Convection

The second mode of heat transfer is called convection. It occurs when a fluid flows over a surface and transfers energy in the form of heat between the fluid and the surface. It can be viewed as conduction in a thin layer near the surface, combined with bulk fluid motion. The transfer of energy depends on the properties of the fluid and occurs due to molecular diffusion and the bulk flow. However, the shape of the surface has a significant impact on the heat transfer [20, p. 5].

Convection can be described by the following relation, where A is the surface area, and T_1 and T_2 are the temperatures of the surface and fluid, respectively. The heat transfer coefficient H represents the strength of convective heat transfer between the surface of the solid and the fluid [20, pp. 5-6].

$$Q = HA(T_1 - T_2) \quad (2.17)$$

There are generally two types of convection and those are forced convection and free, or natural, convection. Forced convection occurs when the flow is driven by external inputs such as pumps or fans. Natural convection occurs when the flow is driven by buoyant forces due to temperature differences within the fluid [20, p. 5].

To describe the relation between the heat transfer across a boundary due to convection and conduction, the non-dimensional Nusselt number can be used. Typically, in laminar flow, the magnitude of heat transfer due to conduction and convection is similar, therefore the Nusselt number is close to one. In a turbulent flow, the magnitude of the heat transfer due to convection is much larger, which yields a Nusselt number often in the range of 100 to 1000 [21].

2.5.2.1 Heat Transfer Coefficients

To model the heat transfer for turbulent flows in *Star-CCM+*, typically, a standard wall function is used, which can be written as follows.

$$\dot{q}_w'' = \frac{\rho_f(y_c)C_{p,f}(y_c)u_*(y^+(y_c))}{T^+(y^+(y_c))} (T_w - T_r) \quad (2.18)$$

The variable y_c is the distance to the near-wall cell center, T^+ is the non-dimensional temperature that depends on the density, ρ and specific heat, C_p of the fluid. From Equation 2.18, the heat transfer coefficient can be written according to Equation 2.19 [22].

$$H(y_c) = \frac{\rho_f(y_c)C_{p,f}(y_c)u_*(y^+(y_c))}{T^+(y^+(y_c))} \quad (2.19)$$

2.5.3 Thermal Radiation

The third mode of heat transfer is thermal radiation. Thermal radiation is emitted by all surfaces at non-zero temperature and is typically associated with wavelengths in the range relevant for thermal emission [20, p. 6]. For radiative heat transfer to take place between two surfaces, they need to have line of sight to each other, and no medium is required between them. Radiation can however, also occur to a surface's surroundings. The heat rate, Q , in W, can be described by Stefan-Boltzmann law seen below [20, p. 6].

$$Q = F_{rad}\sigma A(T_1^4 - T_2^4) \quad (2.20)$$

In the equation above, F_{rad} is a factor that depends on the geometry and surface properties. The variable σ is the Stefan Boltzmann constant with a value of $5.67 \cdot 10^{-8} \text{ W/mK}^4$. The area of the surface that emits or absorbs the energy is given in m^2 . The temperatures T_1 and T_2 are given in kelvin. Due to the fourth-power temperature dependence, radiative heat transfer increases strongly with increasing surface temperature [20, p. 6].

When using a surface-to-surface radiation model in *Star-CCM+*, it requires the computation of radiation patches in order to compute view factors. Patches are created on the surfaces included in the model, and view factors are then calculated by tracing beams between these patches [17].

2.6 Thermal Effusivity

The thermal effusivity e is a property that describes how readily a material can exchange heat with its surroundings and can be calculated according to the following equation.

$$e = \sqrt{\kappa \cdot \rho \cdot C_p} \quad (2.21)$$

It is based on the thermal conductivity κ , density ρ and specific heat C_p . A high value implies that the material can absorb high amounts of energy quickly, while a low value implies the opposite [23, p. 58].

2.7 Discretization

In computational fluid dynamics, meshing is a critical step in the simulation process. It is the process of dividing a geometric domain into smaller elements in which the governing equations are solved. When using the finite volume method, the equations are solved in the center of each element, and in the finite element method the equations are solved in the nodes or edges of the elements. A mesh can be conformal or non-conformal. In a conformal mesh, adjacent parts share matching nodes and element edges [24].

2.7.1 Surface Wrapper

The domain where the equations are solved is called the simulation domain, and the aim of the meshing operation is to create a suitable simulation domain. When a more complex geometry is used, it can be difficult for the mesh to capture the domain properly. For those cases, the *Surface Wrapper* is used to create a closed, manifold and non-intersecting surfaces. Examples of when the *Surface Wrapper* is used are geometries with intersecting parts, surface mismatches, and a high level of detail [24].

2.7.2 Prism Layer

To improve the accuracy of the flow simulation, the prism layer can be used. It is used together with a core volume mesh and creates orthogonal prismatic cells alongside a wall or boundary. There are four important parameters that define the prism layer. The first parameter, defines the overall thickness of the prism layers. Then there is a parameter that defines how many cell layers should be used. The third parameter controls the method used for the thickness distribution of the cells in the prism layers. The last parameter determines the formula used to generate the distribution of the cell layer thickness by stretching. It can either be by geometric progression or a hyperbolic tangent [24].

The prism layer enables the flow to be resolved near the walls, increasing the accuracy of the simulation since it enables forces and heat transfer to also be resolved near the walls. The prism layer also helps to resolve features such as pressure drop or drag. To obtain an accurate prediction of these features, it is important to resolve the temperature and velocity gradients normal to the wall. In the viscous sub-layer of a turbulent boundary layer, these gradients are much larger compared to if they were captured in a coarse mesh. The viscous sub-layer can therefore be resolved directly if it is supported by the turbulence model by using the corresponding y^+ value for the first cell in the prism layer [24].

2.7.3 Thin Mesher

When parts or regions contain thin volumes, the *Thin Mesher* can be used. It recognizes areas of the geometry that are either bulk or thin automatically. For the

thin areas, the *Thin Mesher* creates a prismatic type of mesh and in the bulk areas, the core volume mesher is used. The different meshes are then connected with a conformal mesh [24].

By using the *Thin Mesher*, the overall cell quality is improved, while also reducing the overall cell count compared to an equivalent tetrahedral or polyhedral type core mesh. Furthermore, the convergence can be aided in simulations with thin geometries that involve CHT [24].

2.7.4 Surface Remesher

The *Surface Remesher* can be used to improve the overall quality of an existing surface and optimize it for the volume mesh models. Based on a user-specified target edge length, the *Surface Remesher* re-triangulates the surface. It can also be applied at specific part surfaces and boundaries that need localized refinement. Typically, the *Surface Remesher* is used on surfaces generated by the *Surface Wrapper* [24].

2.7.5 Automatic Surface Repair

When different types of geometric problems exist in a geometry as a consequence of the *Surface Remesher* and *Surface Wrapper*, the *Automatic Surface Repair* can be used to correct those. It uses three different metrics to examine if automatic repair is needed. These are *Surface Quality*, *Surface Proximity* and *Pierced Surfaces*. It fixes those metrics by either re-meshing the surface or deletes the local surface and uses the hole filler [24].

2.7.6 Polyhedral Mesher

The *Polyhedral Mesher* is used to generate a volume mesh containing polyhedral-shaped cells. The polyhedral mesher works well for analysis of heat transfer, swirling flow and complex flow. An advantage of the polyhedral mesh is that it is relatively efficient and easy to build, and requires fewer elements compared to other meshers, while retaining accuracy [24].

2.8 Conjugate Heat Transfer

To model the coupled heat transfer between a solid and a fluid, CHT is used. It combines a fluid-flow model with a thermal model. An illustration of the process can be seen in Figure 2.2.

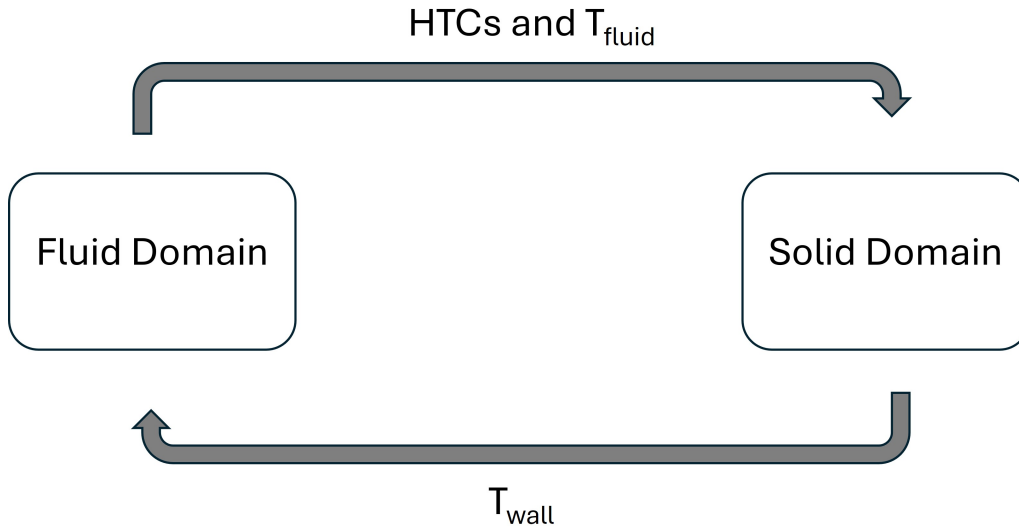


Figure 2.2: Illustration of a conjugate heat transfer coupling approach

In the fluid domain, the convective HTCs on the surfaces of the solid parts and the temperature of the fluid next to these surfaces T_{fluid} are calculated. These quantities allow the convective heat transfer at the solid surfaces to be calculated. The solid domain uses these values to calculate its temperatures. The surface temperatures T_{wall} are then returned to the fluid domain allowing the flow to be recalculated [25].

2.8.1 Energy Coupling

At the interface between a solid and fluid domain in *Star-CCM+*, either implicit or explicit energy coupling can be selected. In the case of implicit energy coupling, the interface is updated every time step if there exists a relative motion between the fluid and the solid domains. For the explicit energy coupling, the same interface established during solution initialization is retained for the complete simulation. However, it does not take into account the physics due to rotation of the solid [26].

2.9 Computational Scalability

When increasing the computational resources by adding cores, a non-linear efficiency curve is achieved. This means that when doubling the number of cores, the computation time is not halved. The loss in efficiency at high core counts is mainly caused by hardware bottlenecks, primarily memory bandwidth limitations and interconnect latency. There is an optimal number of cores for a specified simulation setup which depends directly on the mesh size [27]. The cores are often connected in nodes where a node contains a specified number of cores.

The efficiency is dependent on both CFD software and the type of High-Performance Computing (HPC) cluster but the general optimal setup spans from 50,000 to 100,000 cells/core. Below this range, an almost linear efficiency curve can be expected. If computational speed is more important than efficiency, this ratio can

be reduced to 20,000 cells/core where the efficiency reaches 80% and the speed is maximized [28]. When increasing the number of cores beyond a certain limit, the communication between the cores takes more time than the actual computation, thus turning out to be counter productive.

3

Methodology

This chapter first describes the braking cycles used in the simulations and the experimental rig. It presents the experimental and numerical setups, followed by the investigations conducted using these setups.

3.1 Braking Cycles

To evaluate the performance of the simulations and the effects of different simulation parameters and setups, several braking cycles were used. For most braking cycles, the inlet velocity was fixed at 7.5 m/s, which resulted in a mass-flow-averaged velocity of 2.5 m/s at a plane just upstream of the brake system. The radius of the wheel was simulated as 0.386 m.

3.1.1 Disc Crack Test

The disc crack test cycle consists of repeated braking events of 40 seconds, each with a torque of 1200 Nm per brake with inspections in between to monitor crack formation. A torque of 1200 Nm per brake is high, making this a severe brake event.

The first braking cycle that was used for the thesis was a short and modified cycle of the disc crack test introduced to evaluate different setups and settings of the simulations. It consisted of 40 seconds of braking followed by 60 seconds of cooling and was therefore called the 40/60 braking cycle. This test cycle was used when the accuracy and simulation time were analyzed. The disc rotated with an angular velocity representing a vehicle moving with a constant velocity of 85 km/h = 23.611 m/s resulting in a heat source of 73402 W, calculated from Equation 2.7. After the brake event, the rotation of the disc and the default inlet velocity were kept constant with the same values as during braking.

3.1.2 Alpine Descent

The alpine descent braking cycle represents a truck descending a mountain. Without braking, the truck would accelerate down the slope, but the applied brakes keeps the truck moving with constant velocity. After a period with constant velocity and braking torque, the vehicle is stopped to cool down. This results in a braking cycle with less braking torque than the disc crack test but during a longer time, which still generates high temperatures in the brakes.

3.1.2.1 The 900/1800 Braking Cycle

For this investigation, a modified version of the alpine descent braking cycle was used, called the 900/1800 braking cycle. In this cycle, the brakes were applied for 900 seconds followed by 1800 seconds of cooling. During the cooling phase, the disc rotation was stopped, while the default inlet velocity was kept constant. Since the air flow was maintained during cooling, this did not represent a classical alpine descent braking cycle. The air flow was kept constant because the experimental rig was not designed to withstand the heat that would follow with stagnant air surrounding a hot brake system.

For this simulation, the applied torque was calculated to 306 Nm using the equations described in Section 2.1.1 corresponding to a heat source of 7927 W. The torque was calculated for a truck with four brakes and a mass of 4000 kg descending a slope of 10% at a velocity of 10 m/s. The drag coefficient C_d was set to 0.65 [29], the frontal area to 7.5 m², and the air density to 1.225 kg/m³. The rolling resistance coefficient C_{rr} was set to 0.009 [30] and F_{other} was estimated to around 200 N. All parameters in these calculations were estimated using generic values for quantities. Since this cycle was used only for comparison with other simulations and not with a real truck, it was considered sufficient to use estimated values.

3.1.2.2 Rig Braking Cycle

Due to the rig's operational limits, which are presented in Section 3.2.3, a braking cycle with a lower torque had to be designed. This resulted in a braking cycle similar to the 900/1800 cycle but with a torque of 197 Nm during 333 seconds, followed by a cooling phase of 2000 seconds with the stationary disc. The inlet velocity was 8.0 m/s throughout the cycle and the simulated vehicle velocity was 29.05 m/s corresponding to 104.58 km/h during the braking phase. This cycle was developed by identifying feasible operating conditions in the rig while keeping the torque constant.

3.2 Experimental Setup

In this section, geometry, measurements, and operating limitations of the brake rig built at Chalmers University of Technology are described.

3.2.1 Geometry

The geometry of the brake rig can be seen in Figure 3.1, both in a zoomed-in and zoomed-out view. In the zoomed-out view, the enclosure is visible with its inlet on the left side and its outlet on the right side of the brake system. In the zoomed-in view, the brake system is shown. The full brake system was included except for the hub. The actuator was mounted on the outside of the enclosure behind the black rubber plate located behind the caliper. Instead of the hub, a measuring device was mounted in the front of the disc, with a holder that held the device and its cords.

Behind the brake system, a caliper holder was mounted with an axle in the middle that rotated the disc.



Figure 3.1: Geometry of the experimental brake rig

The rig was rotated clockwise by an electric motor, including an inertia generator, that was designed for a car brake test rig. A fan circulated air through the rig, and a Diesel Particulate Filter (DPF) was used upstream of the inlet to produce a uniform flow. The brakes were applied by applying force on the pads and controlled by a computer program developed for this rig. In this program, torque control was implemented to maintain a near constant torque over time. This program used a Proportional-Integral-Derivative (PID) controller to adjust the brake pressure. To apply the brakes, a piston stroke was generated by pressurized air, which pushed the pads onto the disc. The disc rotation and the inlet velocity were also controlled.

3.2.2 Measurements

In this rig, temperature at different locations and particle emissions were measured. Particle emissions were measured using an Electric Low Pressure Impactor (ELPI) [31].

For the temperature, six thermal probes were mounted in the disc and the locations of the probes are shown in Figure 3.2. The probes were labeled T1 to T6. In the axial direction, T1 and T2 were located at the center of the disc while T3-T6 were located 5 mm from the front side. In the radial direction, T1 and T2 were located at the center of the pads, T3 and T4 close to the inner radius. T5 and T6 were located close to the outer radius. In theory, the pairs T1/T2, T3/T4 and T5/T6 should give the same results during rotation. T6 was not functioning properly and was therefore not monitored.

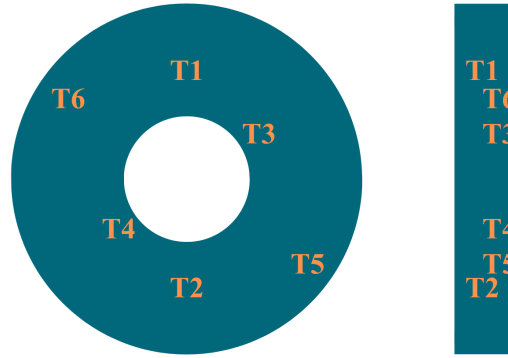


Figure 3.2: Locations of temperature sensors in the disc

To improve control of the braking process, additional measurements were taken including brake pressure, brake torque, inlet velocity, disc angular velocity and inlet temperature. Together with the temperature measurements and time, these values were saved for later comparison with the simulation results.

3.2.3 Operating Limits

Because the rotational motor was designed for car brake testing, its power was insufficient for the larger truck brake to perform the braking cycles originally planned. When the prescribed torque was applied, the motor was unable to maintain disc rotation. This limitation of torque was seen above 200 Nm at which point the disc started to decelerate and eventually came to a stop. Consequently, a new braking cycle was designed, as described previously.

Another limitation identified early in the study was that the air velocity generated by the fan was limited to a maximum of 8 m/s. As a result of this limitation, most simulations were performed with an inlet velocity of 7.5 m/s except the simulation used for comparison with the rig, in which the fan operated at maximum power.

3.3 Numerical Setup

In this section, the base setup of the numerical simulation is described and the choices made are motivated. Modifications to the base setup resulted in several additional configurations, which are also described in this section. These configurations represented different approaches to modeling rotation, radiation, and energy coupling.

3.3.1 Geometry

The simulation geometry followed that of the rig, with some simplifications and modifications. Several differences can be identified when comparing Figure 3.1 with

Figure 3.3. Compared with the numerical model, the experimental rig also included thermal-probe cables, a connection between the measurement device and its holder, and a rubber plate behind the caliper. In addition, the enclosure contained two covered openings in the front and at the top of the enclosure which may have affected the flow. The enclosure was created in *Catia v5* based on a drawing of the standardized rig [32] and can be seen in Figure 3.3 together with an overview of the brake system. The inlet pipe was 0.5 m long, while the outlet pipe was 2 m long. The inlet was designed to represent the rig inlet to capture the boundary layer development.

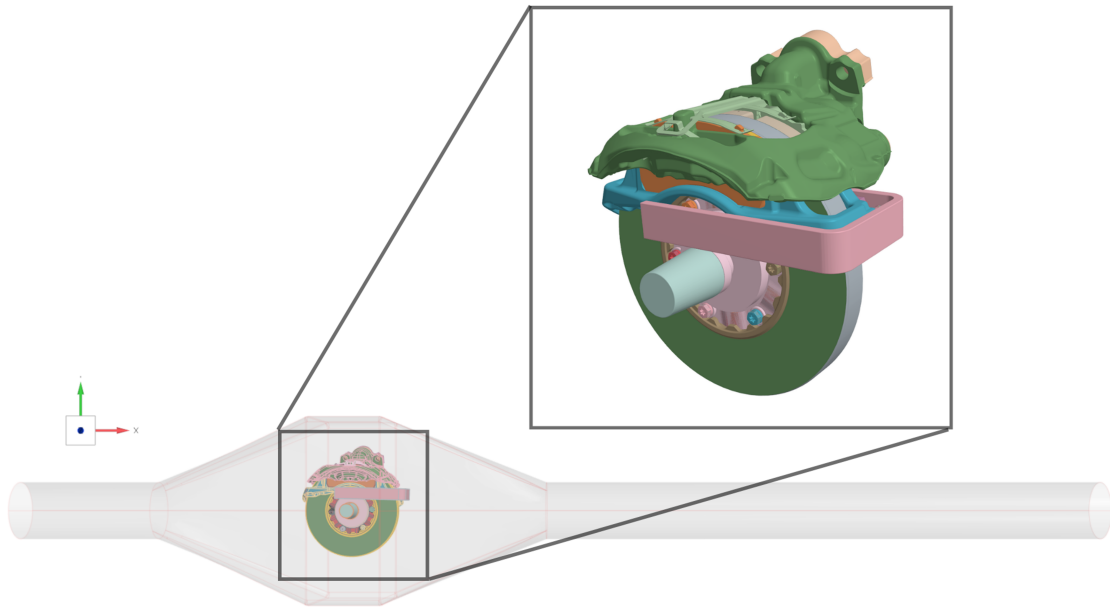


Figure 3.3: Geometry of enclosure and the brake system in the numerical setup

The parts in the brake system are shown in greater detail in Figure 3.4 and 3.5 where the most critical parts are numbered. Different colors represent different parts. The numbered parts are listed below:

1. Caliper holder
2. Axle
3. Disc
4. Thermocouple measuring device
5. Caliper
6. Measuring device holder
7. Pads

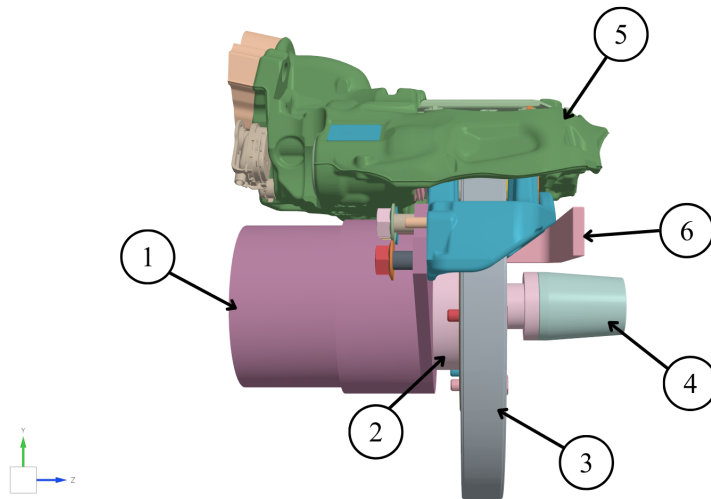


Figure 3.4: Brake system in numerical setup - left view

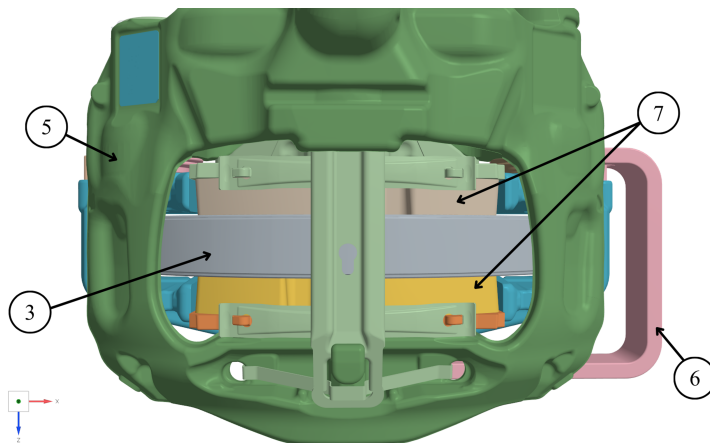


Figure 3.5: Brake system in numerical setup - top view

The air domain was generated by applying a surface wrap to the enclosure and brake system as shown in Figure 3.6. In this figure only the surfaces of the air domain are visible.

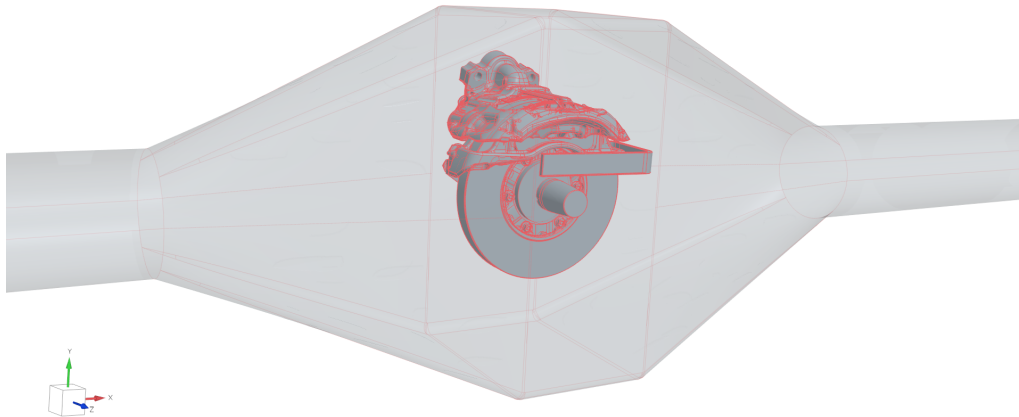


Figure 3.6: Surfaces of the wrapped air domain

3.3.2 Mesh

For the mesh, the following meshers were used:

- Surface Remesher
- Automatic Surface Repair
- Polyhedral Mesher
- Thin Mesher - only in solids
- Prism Layer Mesher - only in fluid

The mesh was generated using the default controls listed in Table 3.1, which were suggested to be used from an industry professional at Volvo. Beyond these settings, custom controls were used to refine the mesh in both the solid and fluid domains where a finer resolution was considered necessary. Refinements were applied to capture gradients and physics.

Table 3.1: Mesh default controls

Setting	Value
Base Size	15 mm
Target Surface Size	50% relative to base
Minimum Surface Size	10% relative to base
Surface Growth Rate	Slow
Number of Thin Layers	3
Volume Growth Rate	1.15
Number of Prism Layers	4
Prism Layer Stretching	1.5
Prism Layer Total Thickness	2 mm

3.3.2.1 Surface Mesh

In Figure 3.7, the surface mesh for the air domain can be seen, and in Figure 3.8 for the solid domain. These two meshes were not conformal to each other and therefore communicated through interpolation rather than direct node-to-node mapping.

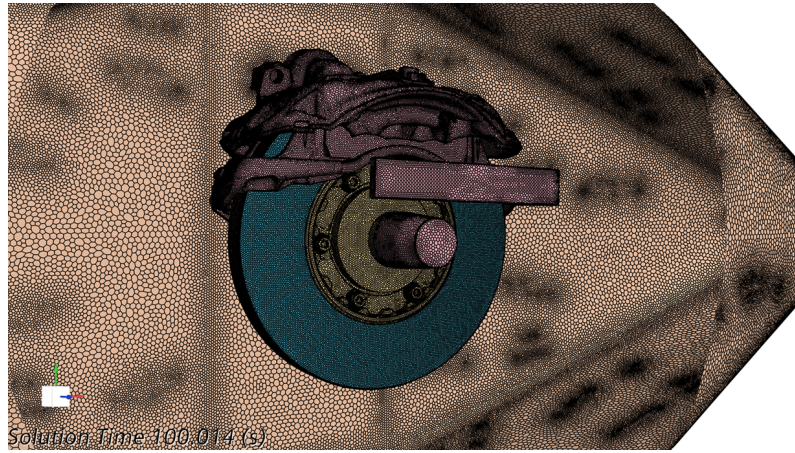


Figure 3.7: Surface mesh of the air domain

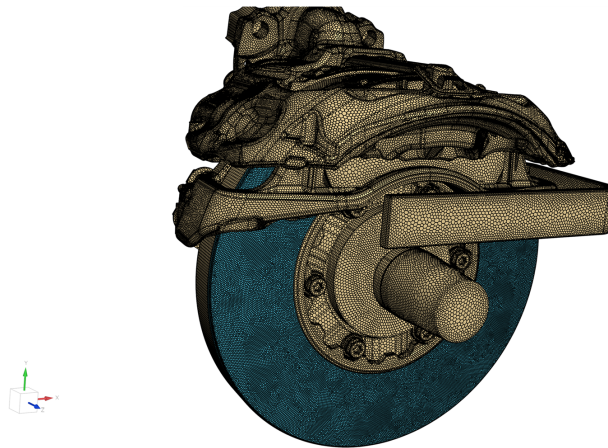


Figure 3.8: Surface mesh of the solid domain without the enclosure

3.3.2.2 Volume Mesh

Using the default and custom controls, together with the surface mesh shown previously, a volume mesh was generated for both the air and solid domains. A cut of the volume mesh for the air domain is shown in Figure 3.9, while a cut of the volume mesh for the solid domain is shown in Figure 3.10. At the disc surface, five prism layer was used instead of the default value of four, using custom controls to achieve the desired y^+ value, which can also be seen in Figure 3.9. To the right of the brake system and close to its surface, the mesh was refined to better capture the flow around it.

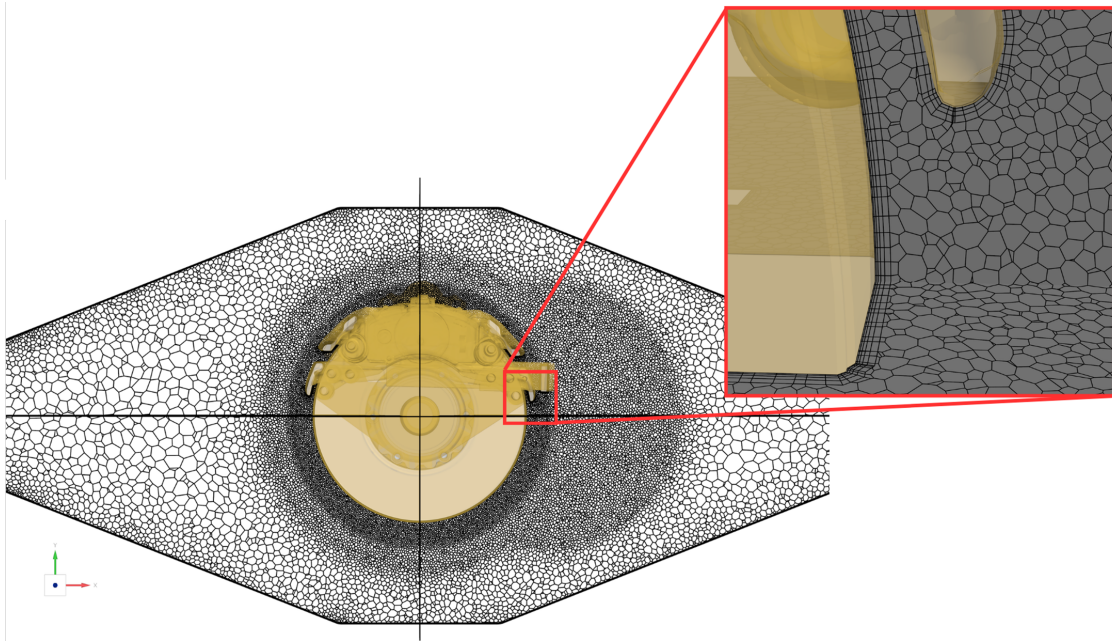


Figure 3.9: Cut of volume mesh of the air domain with zoomed in figure, showing the prism layers

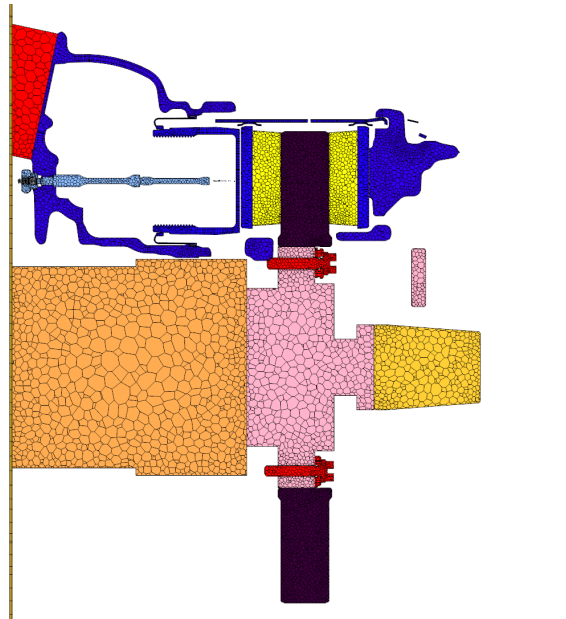


Figure 3.10: Cut of volume mesh of the solid domain - left view

Combining the solid and fluid domain, the cell count for this mesh reaches around 6.335 millions with 1.685 million faces.

3.3.2.3 y^+

To ensure that the boundary layer was resolved, the target y^+ should ideally be either greater than 30 or less than 5. For this model, less than 5 was chosen. See

Figure 3.11 for the y^+ values for the setup with the conditions of the 40/60 braking cycle. The final mesh was iteratively refined by adjusting the number of prism layers and thickness of the prism layer through custom controls to achieve the desired y^+ . It can be seen that in some small areas on the caliper, the y^+ reached values above 5. Although this was not optimal, it was considered acceptable because the caliper was expected to remain at a lower temperature compared to the disc and pad, and therefore, have less influence on the overall brake temperature.

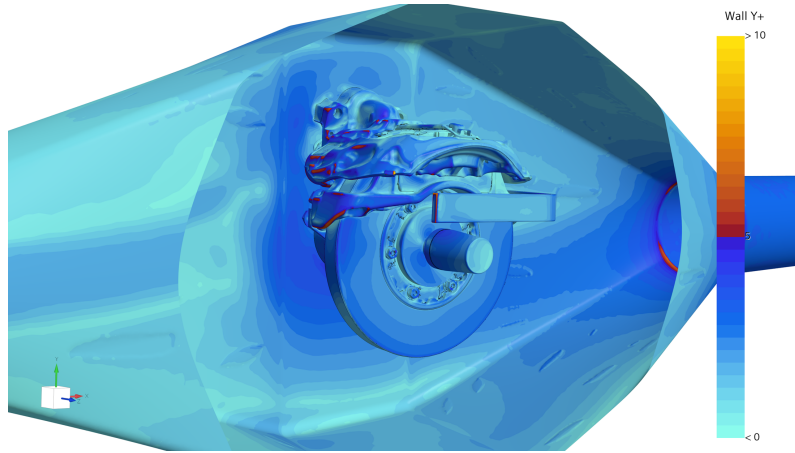


Figure 3.11: y^+ scene of the numerical setup

3.3.3 Physics

The physics models used for the air domain can be seen in the following list in alphabetical order.

- *Gas*
- *Gradients*
- *Ideal gas*
- *K-Epsilon Turbulence*
- *Realizable K-Epsilon Two-Layer*
- *Reynolds-Averaged Navier-Stokes*
- *Segregated Flow*
- *Segregated Fluid Temperature*
- *Solution Interpolation*
- *Steady*
- *Three Dimensional*
- *Turbulent*
- *Two-Layer All y^+ Wall Treatment*
- *Wall Distance*

For the solid domain, the following physics models were used.

- *Gradients*
- *Implicit Unsteady*
- *Multi-Component Solid*

- *Multi-Part Solid*
- *Polynomial Density*
- *Segregated Solid Energy*
- *Solution Interpolation*
- *Three Dimensional*

For both domains, the radiation models were implemented separately in different setups that are explained later. The implemented radiation models were:

- *Gray Thermal Radiation*
- *Radiation*
- *Surface-to-Surface Radiation*
- *Surface Materials*
- *View Factors Calculator*

3.3.3.1 Flow Solver

The air was simulated using a steady RANS solver in order to reduce computational cost. The primary results of interest were the temperature in the brake system rather than the transient development of the flow. Therefore, a steady flow approach was considered sufficient to capture the important physics. The main purpose of the flow simulation was to calculate HTC's and fluid temperatures at the solid boundaries in order to obtain accurate convective heat fluxes and, consequently, accurate temperature predictions.

3.3.3.2 Turbulence

The *Realizable K-Epsilon Two-Layer* turbulence model was employed due to its robustness and computational efficiency for high Reynolds number flows, even if it could result in reduced accuracy when modeling effects near walls. Given the focus of the study on brake thermal behavior, this trade-off was considered acceptable.

3.3.3.3 Rotation

In this thesis, two different approaches for simulating rotation were used. The first was RBM where the rotating parts were physically rotated in the simulation. This was the more accurate method but it came with a higher computational cost compared to the second approach that used CVO.

In the air domain, rotation was simulated by applying a TVS at the boundary of the rotating parts. Both the RBM and the CVO setups used this rotational model in the air domain.

3.3.3.4 Regions

For the RBM configuration of the simulation, three regions were used: a fluid region containing the air, a static solid region and a rotating solid region. In Figure 3.12, the two solid regions can be seen where the yellow parts are the rotating region and

blue parts the static region.

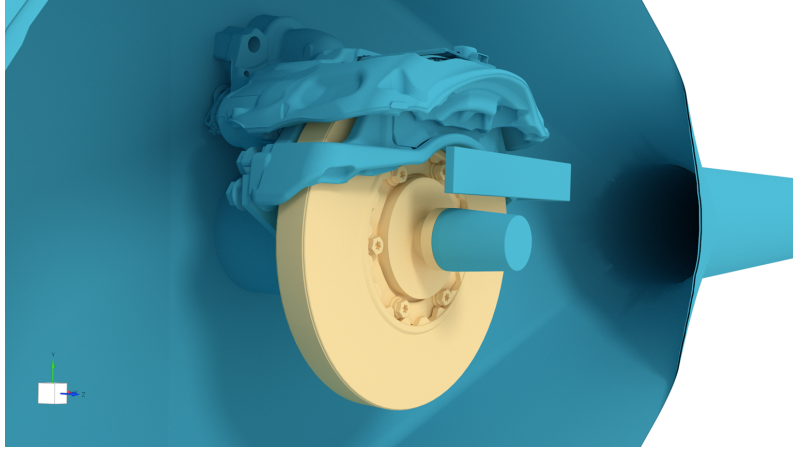


Figure 3.12: Solid regions for the RBM setup

For the CVO configuration, three regions were also used: a fluid region, a convective solid region containing two thin slices of the disc and a solid region containing all other solid parts. The two solid regions are static and can be seen in Figure 3.13 where the yellow parts are the convective region and the blue parts, the remaining solid region.

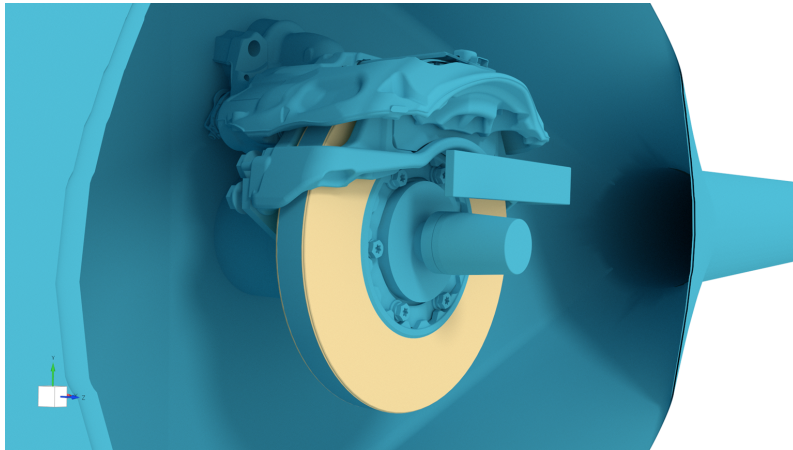


Figure 3.13: Solid regions for the CVO setup

3.3.3.5 Radiation

To model radiative heat transfer inside the system, a *Surface-to-Surface* radiation model was implemented. This approach is suitable for enclosed configurations in which the fluid is assumed to be non-participating. With this radiation model, the standard choice is to use *Gray Thermal Radiation* and to solve the energy balance

correctly, the *View Factors Calculator* was also implemented. This corresponded to a standard setup for surface-to-surface radiation modeling in CFD software [33].

Even though this radiation model setup was considered relatively computational efficient, the combination of this and RBM drastically increased the computational cost. Each time the radiation model was updated, RBM had moved the rotating parts, which required the radiation patches and view factors to be recalculated.

Two different implementations were used, one with radiation enabled in the fluid domain and one in the solid domain. For the one in the fluid domain, the radiation was enabled internally to radiate to and from the different boundaries of the domain. In the setup using radiation in the solid domain, the radiation was enabled externally to radiate out of the domain and between the different solid regions.

Radiation in the fluid domain was computationally economical for this type of simulation because new patches and view factors were only calculated when the fluid domain was solved. For a steady flow, as used in this simulation, this process could be manually controlled, making it less computationally expensive than a simulation using radiation in the solid domain. When radiation was applied in the solid domain in an unsteady time scale, the radiation had to be calculated for every time step.

When CVO was used instead of RBM, no mesh motion was introduced, and the radiation patches and view factors therefore only had to be calculated once during initialization.

To further increase the computational efficiency when including radiation models, the patches created on the enclosure walls were limited in size. The *Patch/Face Proportion* was set to 0.5 which, combined with a bigger surface size than the brake system's, resulted in around 975 patches for the whole enclosure. This value was increased because capturing local radiation effects on this wall was considered less important, while capturing the general effect of its inclusion was still deemed necessary.

3.3.3.6 Energy Coupling

For the simulations used in this project, the implicit energy coupling was chosen because the loss of interface was deemed less significant compared to the loss of physics associated with the explicit approach.

3.3.3.7 Boundaries

The inlet of the enclosure was set to a velocity inlet, while the outlet was set to a pressure outlet. At the inlet, the velocity was set to a constant value to approximate the rig conditions, where the DPF in the experimental rig helped produce a more uniform inflow. The pressure outlet was chosen to drive the pressure field and the outlet was long enough so that it would not affect the flow next to the brake system. The walls of the enclosure were set to a wall boundary with a convective outer

boundary. At the outer boundary, the HTC was set to $8 \text{ W/m}^2\text{K}$ with an external temperature of 23°C .

3.3.3.8 Heat Application

The heat was applied in the interface between the pads and the disc, see Figure 3.14. For the braking cycles that were tested, this heat was constant in the braking stages and zero in the cooling stages. In more complex braking events, this could vary over time. The heat was applied at an interface that combined both sides of the disc, implying identical heat generation on both sides.

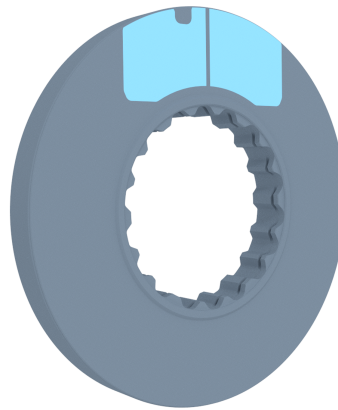


Figure 3.14: The interface between brake disc and brake pad where the heat is applied

3.3.3.9 Materials

Different materials were used to simulate different parts of the brake system. The different material properties are listed in Table 3.2 below. In the table, some values are given over a temperature range. In this temperature range, the property varied from the first to the second value but was not always linear. In other words, some properties were represented using higher-order polynomial fits. Outside this range, the property was treated as constant, taking the boundary value of the range. For the thermophysical properties for which temperature dependent data were not available, a constant value was assumed.

Table 3.2: Material properties of the used materials

Material	Conductivity [W/mK]	Specific Heat [J/kgK]	Density [kg/m ³]
Grey Iron	45-38 (0-1000°C)	434-600 (0-1000°C)	7212-7020 (0-1000°C)
Steel VSHR420	50	520	7850
Organic Pad	4.9-2.0 (0-300°C)	720-438 (100-249°C)	2400
Plastic	0.9	692	1100

The disc and the caliper were modeled as Grey Iron. The material of the pad was modeled as Organic Pad. Some smaller parts inside the caliper and the measuring device were modeled as plastic. The rest of the rig was modeled as steel, including the enclosure, the measuring device holder, the caliper holder, the axle, all bolts and some more smaller parts inside the brake system.

3.3.4 Monitored Variables

This chapter presents the variables obtained from the simulations and outlines the quantities monitored throughout the analysis.

3.3.4.1 Thermal Probes

Point probes were created to measure the temperatures in the same locations as the thermocouples in the experimental rig. These points were fixed in space and therefore, measured along the path generated by the disc rotation. This did not affect the results during rotation because the temperatures in the simulations were assumed to vary smoothly in both time and space. In the simulations the probes were labeled T1/T2, T3/T4 and T5/T6, representing the corresponding probe pairs in the rig, meaning that only one probe in the simulations were compared with two in the rig. These probes were located to the left of the center of the disc when viewed from the front, see Figure 3.15.

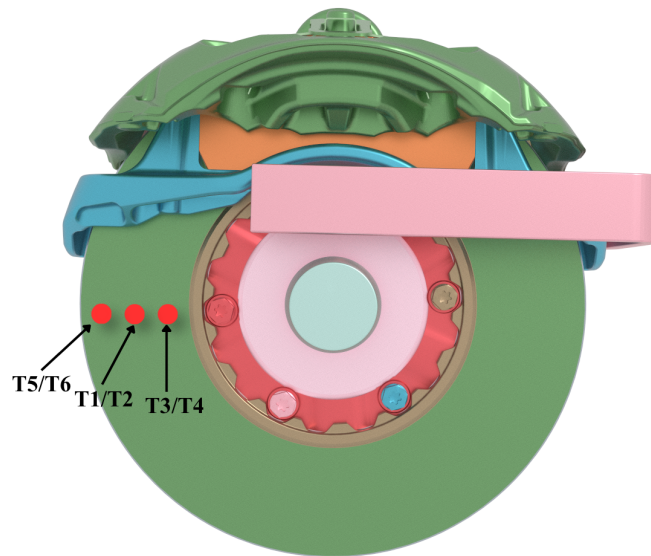


Figure 3.15: Thermal probes placements in the simulations

3.3.4.2 Maximum and Average Temperatures

The maximum and average temperature of different parts and surfaces were also monitored in the simulations. The parts that were monitored were the pads and the disc but only the temperature in the disc was used in comparisons.

3.3.4.3 Heat Rates

To examine how the heat moved inside the rig, the different heat rates were also monitored. These were monitored only on the rotating parts. The heat rates also showed how different modes of heat transfer contributed to cooling the disc. This was done by comparing different types of heat rates, such as radiation, convection and conduction.

3.3.4.4 Computational Time

Computational time was monitored using the total solver Central Processing Unit (CPU) time. This quantity measures the total accumulated core time used by the simulation.

3.3.4.5 Scenes

To get visual representations of the results, some scenes were created. The generated scenes showed temperature, flow velocity, rotational velocity, and supporting quantities such as y^+ and the volume mesh.

3.3.5 Simulation

To simulate the different braking cycles the simulation settings had to be chosen. This included the coupling between the fluid and solid domains, time scales, stages, solver settings and simulation operations. The coupling required to solve a CHT problem was performed as described in Section 2.8. This was done automatically by *Star-CCM+* using interfaces between the domains.

3.3.5.1 Time Scales

Two different time scales were used in the simulations. For the solid parts, a transient time scale was used to capture the heat in the brake system over time. The fluid domain was simulated in a steady time scale. To capture the influence of the increasing brake temperature on the fluid domain, a number of steady-flow iterations were performed at specified time intervals during the simulation. For most simulations, this was set to 10 iterations twice every second.

The time step was set to match a specified angle between the steps which depended on the rotational speed of the disc. This was mainly important for the RBM setup due to how the heat was locally applied. To avoid synchronization between the heat application and the time step which could lead to hot spots, the angular step was set to 23° for the 40/60 braking cycle and 46° for both the 900/1800 and the Rig braking cycle. The angular step was doubled for these braking cycles because the simulations were too long and needed to be shortened in order to be completed. Furthermore, the heat was applied slower, leading to smaller gradients and therefore a larger time step could be used. This resulted in time steps of 0.013125 seconds for the 40/60 cycle, 0.03099 seconds for the 900/1800 cycle, and 0.010668 seconds for the Rig braking cycle. The time step was not doubled for the 900/1800 and the

Rig braking cycle because they had different rotational speeds. The same time steps were used in the matching CVO setups.

During the cooling phase, the time step was increased because no heat was applied to the system and therefore, there was no risk of the time step matching the rotation and affecting the results. The time step was tripled for the first 60 seconds of cooling and increased by a factor of 10 after that. Since the 40/60 simulation ended before the largest time step was reached, this adjustment was only applied to the 900/1800 and the Rig braking cycle.

3.3.5.2 Stages

All braking cycles consisted of two different phases, braking and cooling. Therefore, two stages were implemented in the simulations that represented those phases. In these stages, different parameters could be staged which meant that they could differ between them. One example is that heat was applied in the braking stage but not in the cooling stage.

3.3.5.3 Solvers

Except for the *View Factors Calculator* solver, which used different *Number of Beams* between setups, the solvers were left with default settings.

3.3.5.4 Simulation Operation Base Setup

The base *Simulation Operation* consisted of four parts: initialization, braking, cooling, and final saving of the simulation. The initialization sequence cleared previous results, initialized the solution, and solved the fluid domain to convergence before the transient solid solution was started. The removing of invalid cells was done three times because when removing one time, a few others could appear. This sequence consisted of the following operations.

1. Mesh
2. Remove Invalid Cells
3. Clear Solution
4. Initialize Solution
5. Solve Physics - Fluid domain (2000 iterations)

After the initialization, the braking sequence was executed. This sequence was identical to the cooling sequence except the staged parameters and the length of the stage. Both sequences are presented below together.

1. Set Active Stage - Braking/Cooling
2. Outer loop - Until stage is over
 - (a) Solve Physics - Fluid domain (10 iterations)
 - (b) Inner loop - Until solution time increased by 0.5 seconds
 - i. Solve Physics - Solid domain

In these sequences, first, the desired stage was activated which would change all staged parameters. Then, an outer loop was executed which continued until the specified stage was over. Whether a stage was over was based on a specified time limit. Inside this loop, there was another loop that was executed to solve the solid domain. When solving the solid domain, the time increased and every 0.5 seconds, the loop was exited to solve the fluid domain for an additional 10 iterations. When the braking sequence was over, the cooling sequence was executed and when the cooling sequence was over, the simulation was saved.

3.3.5.5 Scalability of Computational Power

Due to the limited time available for this project and the high computational power provided by Volvo Group, the simulations aimed to maximize speed. Therefore, the target was to use close to 20000 cells/core instead of the more efficient span of 50000-100000 cells/core. For the simulations, 384 cores were used for each case, resulting in 16497 cells/core for the RBM setup and a similar value for the CVO setup. It was first thought that when choosing a cells/core ratio, both cells and surfaces should be counted and that is why the targeted cells/core ratio was surpassed. When also including surfaces, 20879 cells/surfaces per core was achieved. To compare the computational cost of the different setups fairly, the number of cores was kept constant across all simulations in order to avoid comparing cases with different parallel efficiencies. Furthermore, adjacent nodes were used for all simulations to avoid affecting the comparison.

3.4 Investigations

The main aim of this thesis was to obtain an accurate simulation model compared to the real brake rig. One key methodological issue was how radiation and rotation should be modeled in such a setup. This motivated a dedicated investigation of radiation modeling approaches and also if CVO could be a suitable substitute to RBM. This section describes the investigations done for the radiation analysis part of the thesis and the CVO analysis part as well as how the final simulation was compared to the real rig.

3.4.1 Effects of Implementing Radiation

Due to the increased computational cost of simulating RBM with radiation, the first question was how much including radiation would affect the disc temperature. This was investigated first in order to determine whether a more detailed analysis of radiation was justified. For this investigation, both the 40/60 and 900/1800 braking cycles were used. To evaluate the effect of radiation, temperatures, heat rates and computational time were compared in plots.

However, a problem was identified when using RBM combined with a radiation model in the fluid domain which led to a second setup in which the radiation model was implemented in the solid domain instead. These two setups were named Setup

A and Setup B, respectively. The simulations executed in this investigation can be seen in Table 3.3.

Table 3.3: Simulated setups for investigating the effects of implementing radiation

Setup	Braking Cycle
A	40/60
A	40/60 (Stopped rotation of disc during cooling)
B	40/60
B	900/1800

3.4.2 Optimized Radiation

Following the analysis of the effects of including a radiation model in combination with RBM, an attempt was made to develop an optimized radiation model to get as accurate results as possible with the lowest computational cost. This was done by first identifying which parameters affected the computational time.

There are different ways to optimize view factor calculations. The first one is to adjust how many beams each patch can trace towards the other patches, *Number of Beams*. A value of 1024 is the default. With a higher *Number of Beams*, the accuracy is increased but a lower value may be sufficient.

A second option is the *Update Frequency*, which controls how often the view factors should be recomputed. By recalculating them less frequently, computational cost can be decreased. By default, it is set to update each time step. However, the view factors solver is semi-intelligent which means that if it finds it necessary for accuracy, it can override the specified frequency. This may occur when the solver detects a mesh change, interface re-imprint or interface reset, making it not applicable to this optimization study [17].

Related to radiation, there is another option in *Star-CCM+* that can have a noticeable impact on the performance of the calculations. It is the *Patch/Face Proportion*. It specifies the fraction of patches allocated to each boundary face. For example, a value of 20 means that 100 faces would be mapped to 20 patches. Usually a value of 3-20 is sufficient where lower values increase computational performance but decrease accuracy [17].

A manual parameter sweep of the *Number of Beams* and *Patch/Face Proportion* was carried out to see the differences in temperatures over time for the 900/1800 braking cycle. The 900/1800 braking cycle was used because it caused the disc to reach higher temperatures over a longer period of time, resulting in a larger difference between the radiation models. See Table 3.4 for the different parameter values of the parameter sweep.

Table 3.4: Parameter values of the manual parameter sweep

	Patch/Face Proportion	Number of Beams
Simulation 1	10	256
Simulation 2	5	128
Simulation 3	2.5	64
Simulation 4	1	64

Computational time was monitored, and the optimized setup was defined as the fastest simulation that still produced temperatures close to those of the most computationally expensive radiation case.

To determine which parameter affected computational time the most, two extra simulations were executed and the parameter values of these can be seen in Table 3.5.

Table 3.5: Parameter values of the extra simulations

	Patch/Face Proportion	Number of Beams
Simulation 5	10	64
Simulation 6	5	64

3.4.3 Convective Velocity Option

Another way to reduce the computational cost associated with radiation was to use a CVO to simulate disc rotation instead of using RBM. This would allow the model to calculate radiation patches and view factors only once during initialization. To test this, simulations were created for both radiation in the solid and the fluid domain. The setup with radiation in the solid domain was simulated to clearly show the effects of a CVO while the setup with radiation in the fluid was seen as more industrially usable. The second setup was considered more industrially relevant because it calculated radiation less frequently while still using a high *Patch/Face Proportion* and *Number of Beams*, which was hypothesized to retain accuracy at lower computational cost.

In total, three different setups were compared in this investigation:

1. RBM with optimized radiation in solids
2. CVO with default radiation in fluid
3. CVO with optimized radiation in solids

They were compared using plots of the temperatures, heat rates and computational time.

3.4.4 Verification of Simulations

Finally, the CVO and RBM simulations were compared with a real test conducted in the rig. The braking cycle used was the Rig braking cycle. In these tests,

the temperatures were evaluated using the values of the thermal probes T1 to T5. For the temperature results from the rig, the temperatures were averaged in each pair except for T5 because its corresponding paired probe T6 was not functioning. Temperature values were compared from three different sources:

1. RBM with optimized radiation in solids
2. CVO with optimized radiation in solids
3. Data from experimental brake rig

4

Results

This chapter presents the results of the investigations. It first covers the implementation and optimization of the radiation model, followed by the CVO setup, and finally the comparison between the experimental rig and the simulations. The results are evaluated based on heat rates, temperatures, total solver CPU time, and visualizations of radiation, temperature, and flow.

4.1 Effects of Implementing Radiation

To see the effects of implementing radiation in the simulation, results were extracted from simulations with and without radiation enabled. These simulations used the RBM setup.

4.1.1 Setup A

In Setup A, the radiation was implemented internally in the fluid domain. The maximum and average temperature of the disc can be seen in Figure 4.1. This setup simulated the 40/60 braking cycle.

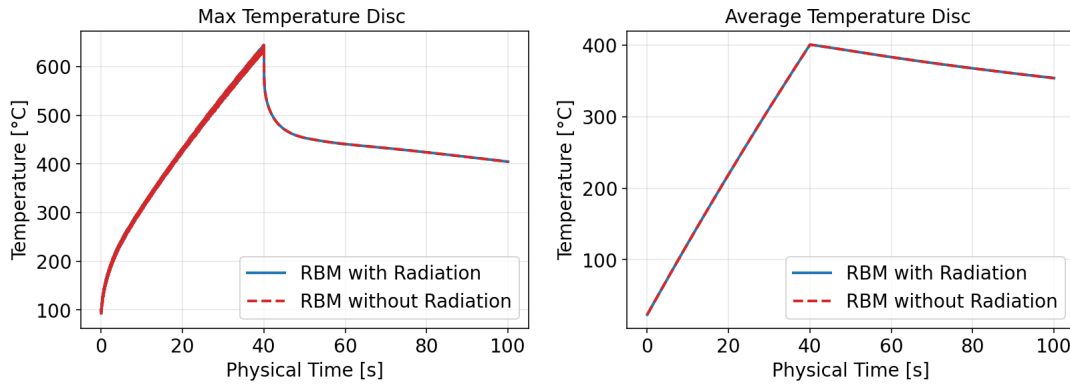


Figure 4.1: Maximum and average temperature in the disc with and without radiation for Setup A, simulating the 40/60 braking cycle

The figures show no noticeable temperature difference between the cases with and without radiation. For the same simulations, the heat rates leaving the rotating parts can be seen in Figure 4.2.

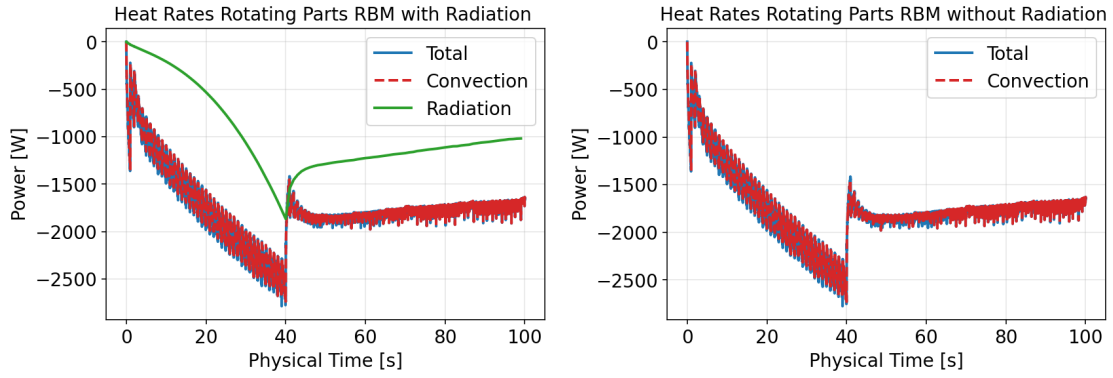


Figure 4.2: The heat rates leaving the rotating parts for Setup A, simulating the 40/60 braking cycle

These plots show that the radiative heat rate is not included when calculating the total heat rate. The total heat rate should be the sum of the convective and radiative heat rates. When comparing the left plot with the right plot, which shows the simulation without radiation, it can also be seen that the total heat rate over the boundary is identical in both plots.

The total solver CPU time for the same simulations can be seen in Figure 4.3. It is possible to see that the simulation without radiation requires less CPU time. In fact, the total solver CPU time decreases by 20.3% without radiation.

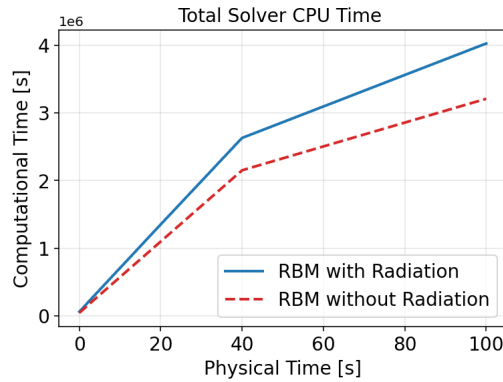


Figure 4.3: The total solver CPU time for Setup A with and without radiation, simulating the 40/60 braking cycle

One modified simulation was also performed; in this case, the disc was stationary during the cooling phase. The heat rates leaving the disc is shown in Figure 4.4 which shows that when the disc stops rotating, the radiative heat rate is added to the total heat rate.

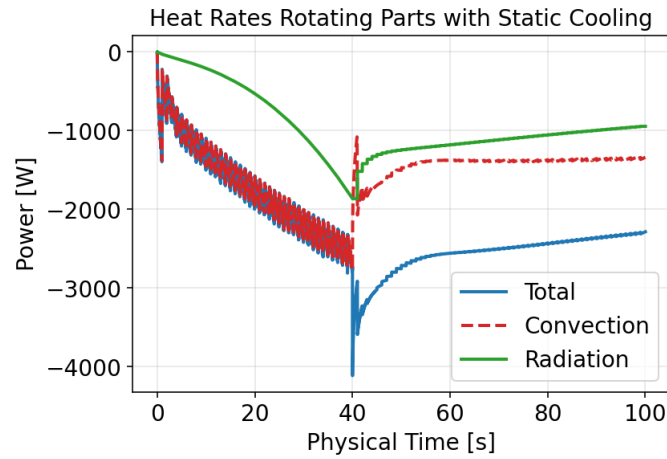


Figure 4.4: Heat rates leaving the rotating parts with static cooling for Setup A, simulating the 40/60 braking cycle

This indicates that, in Setup A, radiation implemented in the fluid domain does not affect the energy balance as expected when combined with RBM. The problem was documented and reported to the software support.

4.1.2 Setup B

In Setup B, radiation was instead implemented in the solid domain. Both the 40/60 and the 900/1800 braking cycle were simulated using this setup.

4.1.2.1 40/60

For the 40/60 cycle, the maximum and average temperature in the disc are shown in Figure 4.5. For this setup, a temperature difference can be observed between the simulations with and without radiation. The difference can be observed in both the heating and the cooling phase.

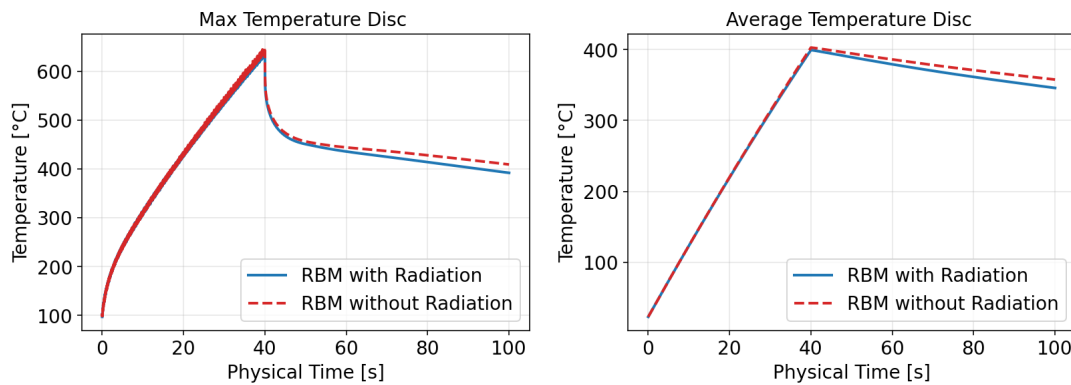


Figure 4.5: Maximum and average temperature in the disc with and without radiation for Setup B, simulating the 40/60 braking cycle

For the same simulations, the heat rates leaving the rotating parts can be seen in Figure 4.6. For this setup, the radiative heat rate is calculated and added to the

convective heat rate to obtain the total heat rate. The total heat rate of the setup without radiation is also equal to just the convective heat rate in the setup with radiation.

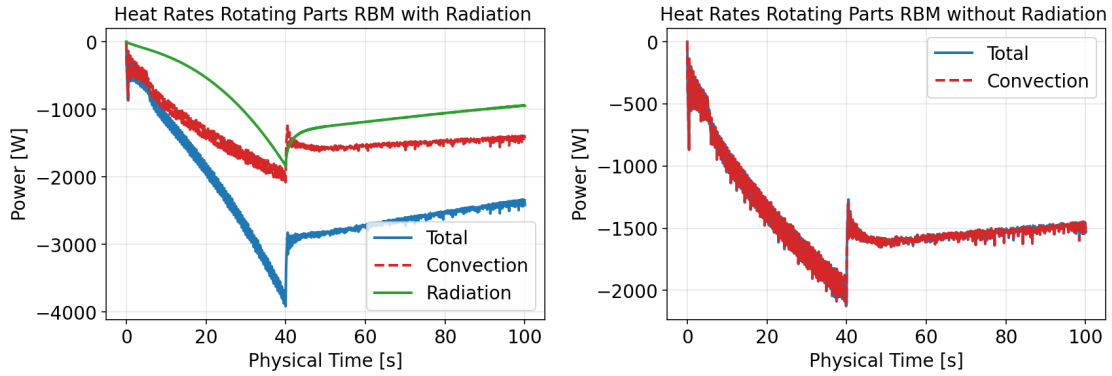


Figure 4.6: Heat rates leaving the rotating parts for simulations with and without radiation for Setup B, simulating the 40/60 braking cycle

The thermal energy applied and the energy leaving the rotating parts are shown in Table 4.1.

Table 4.1: Thermal energy applied to the brake system and energy leaving the rotating parts for Setup B, simulating the 40/60 braking cycle

	Thermal Energy Applied [Wh]	Radiation Rotating Parts [Wh]	Convection Rotating Parts [Wh]
40/60 with Radiation	816	-26	-39

The total solver CPU time for the two cases is shown in Figure 4.7. Excluding radiation reduces the total solver CPU time by 67.9%.

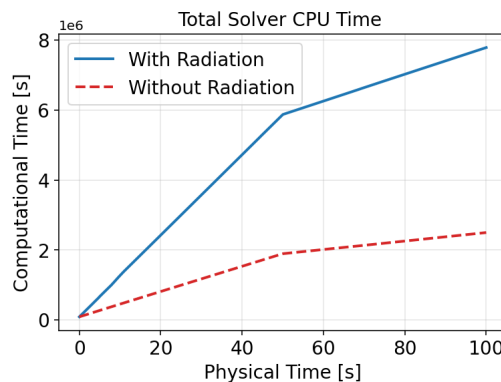


Figure 4.7: Total CPU time for simulations with and without radiation for Setup B, simulating the 40/60 braking cycle

4.1.2.2 900/1800

For the 900/1800 braking cycle, the maximum and average temperatures of the disc are shown in Figure 4.8. The figure shows that the disc temperatures are higher in the simulation without radiation.

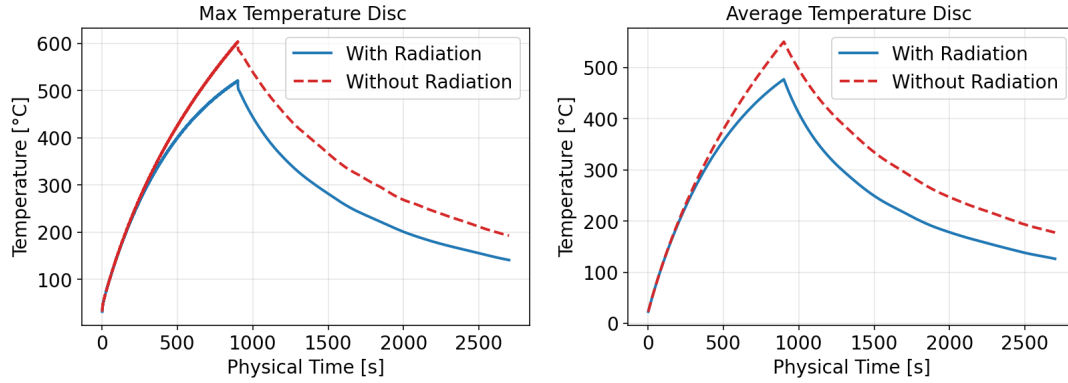


Figure 4.8: Maximum and average temperature in the disc with and without radiation for Setup B, simulating the 900/1800 braking cycle

For the same simulations, the heat rates leaving the rotating parts can be seen in Figure 4.9. The figure shows, as for the 40/60 braking cycle, that with radiation, the radiative heat rate is calculated and added to the convective heat rate to obtain the total heat rate. An oscillating convective heat rate can be seen during the cooling phase.

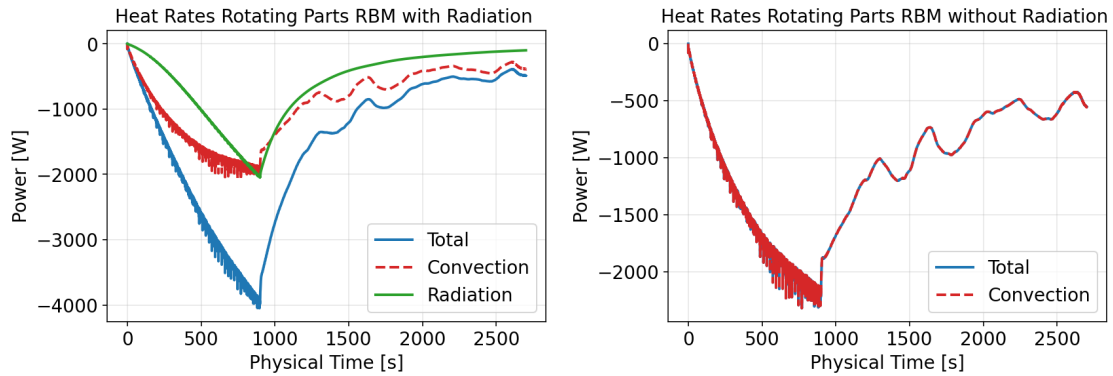


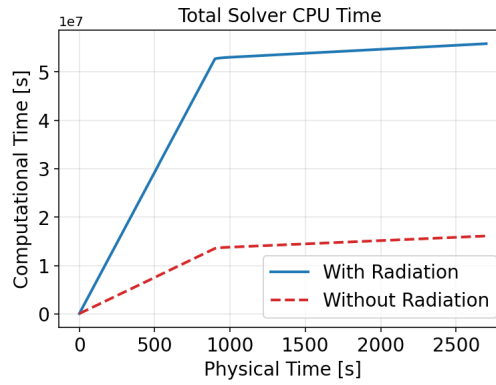
Figure 4.9: Heat rates leaving the rotating parts for simulations with and without radiation for Setup B, simulating the 900/1800 braking cycle

The thermal energy applied and the energy leaving the rotating parts can be seen in Table 4.2.

Table 4.2: Thermal energy applied to brake system and energy leaving the rotating parts for Setup B, simulating the 900/1800 braking cycle

	Thermal Energy Applied [Wh]	Radiation Rotating Parts [Wh]	Convection Rotating Parts [Wh]
900/1800 with Radiation	1982	-447	-667

The total solver CPU time was also plotted for the two different cases, see Figure 4.10. For this braking cycle, excluding radiation, the total solver CPU time decreases by 71.1%.

**Figure 4.10:** Total solver CPU time for simulations with and without radiation for Setup B, simulating the 900/1800 braking cycle

4.2 Optimized Radiation

In this section, the results of an optimization of the radiation settings for the 900/1800 braking cycle are presented. This optimization uses Setup B where the radiation is enabled in the solid domain and the rotation is modeled as RBM.

4.2.1 Simulation 1-4

The maximum and average temperatures in the disc are plotted for the first four configurations of radiation settings in Figure 4.11. The figure shows that the temperature curves are very similar for all four simulations.

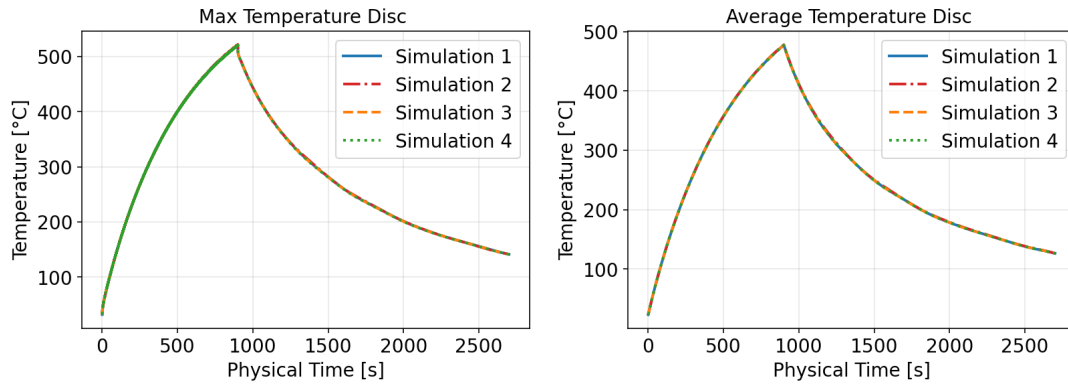


Figure 4.11: Maximum and average temperature in the disc with different radiation parameters for Setup B, simulating the 900/1800 braking cycle

The total solver CPU time for the same simulations is plotted in Figure 4.12 which shows that Simulation 3 and 4 require the least CPU time.

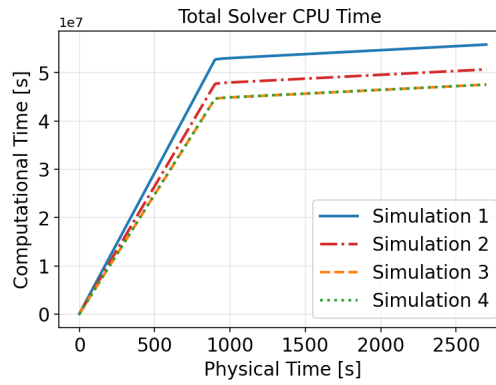


Figure 4.12: Total solver CPU time for simulations with different radiation parameters for Setup B, simulating the 900/1800 braking cycle

The time savings and Root Mean Square Error (RMSE), relative to Simulation 1, are presented in Table 4.3.

Table 4.3: Comparison of the different radiation parameter settings

	Sim 1	Sim 2	Sim 3	Sim 4
Patch/Face Proportion	10	5	2.5	1
Nr of Beams	256	128	64	64
Max Temp Disc RMSE	-	0.41	0.109	0.35
Avg Temp Disc RMSE	-	0.45	0.056	0.21
Time Saved [%]	-	9.2	14.8	14.8

The total, convective and radiative heat rates leaving the rotating parts are plotted in Figure 4.13. Once again, the figure shows that the results are similar for the different simulations.

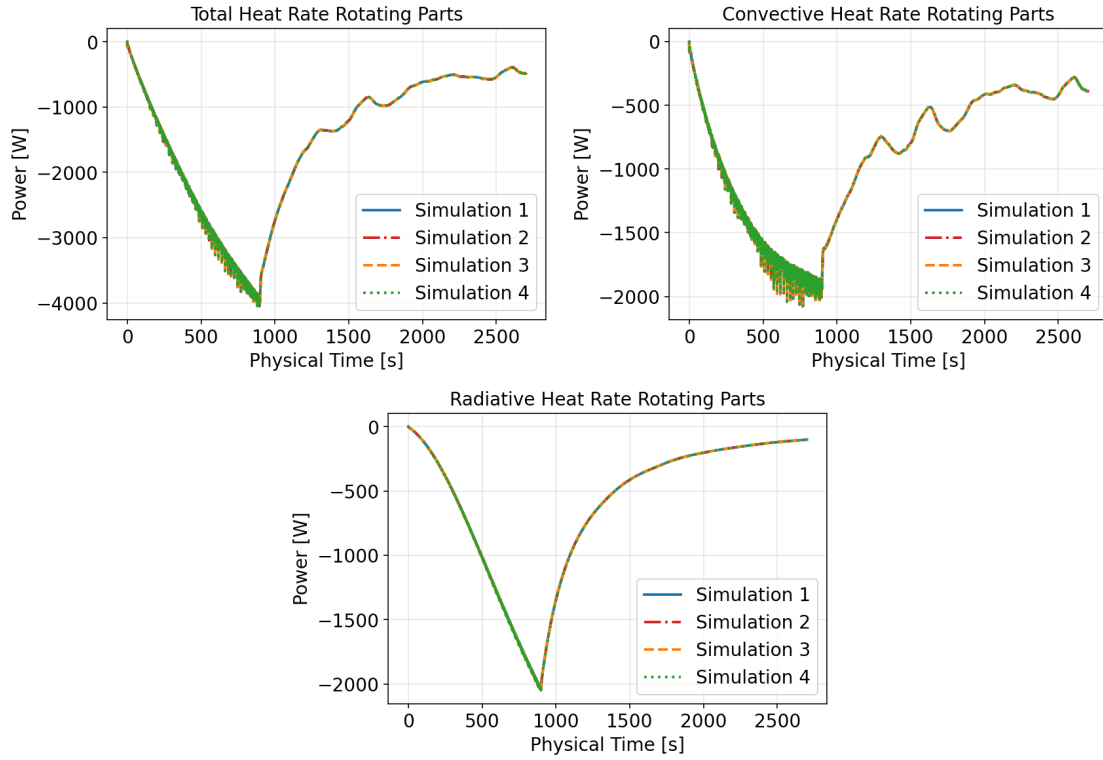


Figure 4.13: Total, convective and radiative heat rates in the rotating parts with different radiation parameters for Setup B, simulating the 900/1800 braking cycle

4.2.2 Simulations 5-6

Simulations 3 and 4 require the same computational time and uses the same *Number of Beams*. To test the importance of the parameters, two more configurations were simulated using that value of *Number of Beams* but a larger *Patch/Face Proportion*.

The total solver CPU time was plotted for the extra simulations. They are compared with Simulations 1 to 4 in Figure 4.14. The figure shows that Simulations 6 and 2 are similar to each other, with different *Number of Beams* but the same *Patch/Face Proportion*. Another observation is that Simulations 3 and 4 are still the fastest.

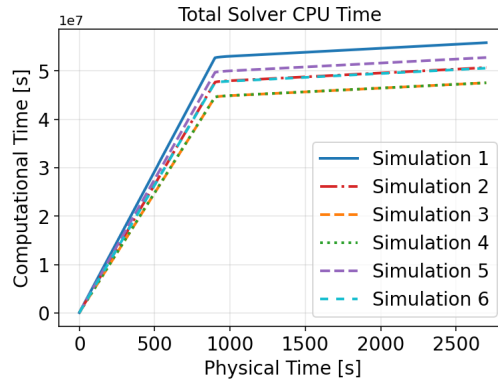


Figure 4.14: Total solver CPU time for all set of radiation settings for Setup B, simulating the 900/1800 braking cycle

For Simulation 5 and 6, the RMSE values and time savings are calculated relative to Simulation 1 and are presented in Table 4.4. It shows that Simulation 6 with a lower *Patch/Face Proportion* has a lower RMSE than Simulation 5 for the maximum and average temperatures compared to Simulation 1.

Table 4.4: The RMSE and time savings of Simulations 5 and 6 for Setup B, simulating the 900/1800 braking cycle

	Sim 1	Sim 5	Sim 6
Patch/Face Proportion	10	10	5
Nr of Beams	256	64	64
Max Temp Disc RMSE	-	0.44	0.42
Avg Temp Disc RMSE	-	0.48	0.47
Time Saved [%]	-	5.5	9.4

Because Simulation 4 used the lowest parameter values while still producing results close to those of Simulation 1, it was selected as the optimized configuration.

4.2.3 Visualization

In this section, some simulation visualizations are presented, showing the radiative heat flux, surface temperature and air flow. This will be presented for both the 40/60 and the 900/1800 braking cycles.

4.2.3.1 40/60

Radiative heat flux was only saved as a time-dependent result for the 40/60 braking cycle and a picture of a scene showing it at 40s, where the temperature and radiative heat flux reach a maximum, can be seen in Figure 4.15.

4. Results

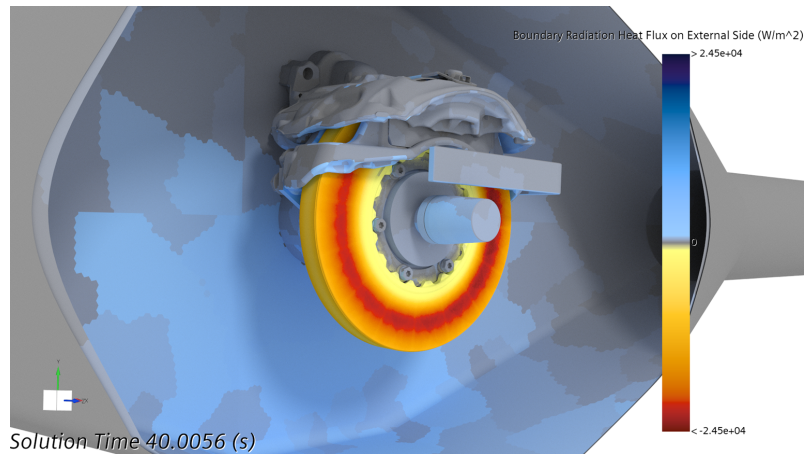


Figure 4.15: Radiative heat flux for 40/60 braking cycle at 40 seconds

The figure shows radiative heat transfer from the disc to surrounding components such as the caliper and enclosure. The disc is radiating out energy while the caliper and enclosure are absorbing it. The size-limited radiation patches on the enclosure can also be seen when inspecting the pattern on the enclosure wall. In the brake system, the smaller patches can also be seen when inspecting the caliper surface.

For the same simulation, the surface temperature of the brake system can be seen in Figure 4.16 for a few different time steps. The time steps presented in the figure are 30, 40 and 50 seconds.

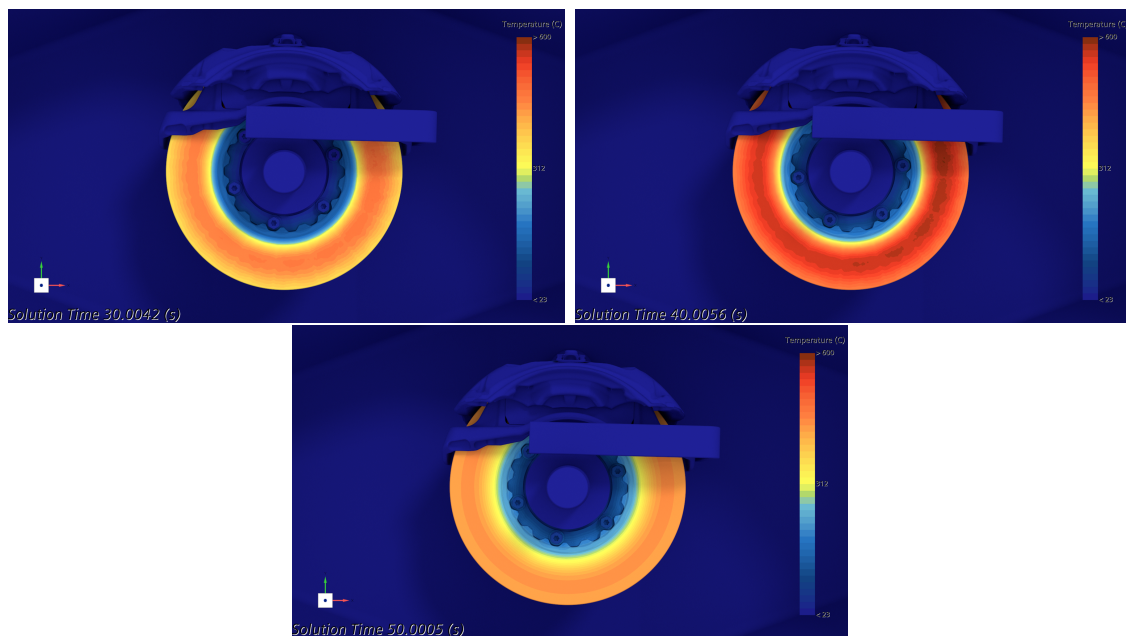


Figure 4.16: Temperature of the brake system for the 40/60 braking cycle at 30, 40 and 50 seconds

It can be seen in this figure that the disc is heated up during the braking stage and cooled down during cooling. Furthermore, a pattern can be seen from the heat

application during braking which is smoothed out during the cooling stage.

The flow was also captured at the last time step, at 100 seconds, and it is presented in Figure 4.17 and 4.18. The first figure shows the flow from the front, while the second shows it from the top.

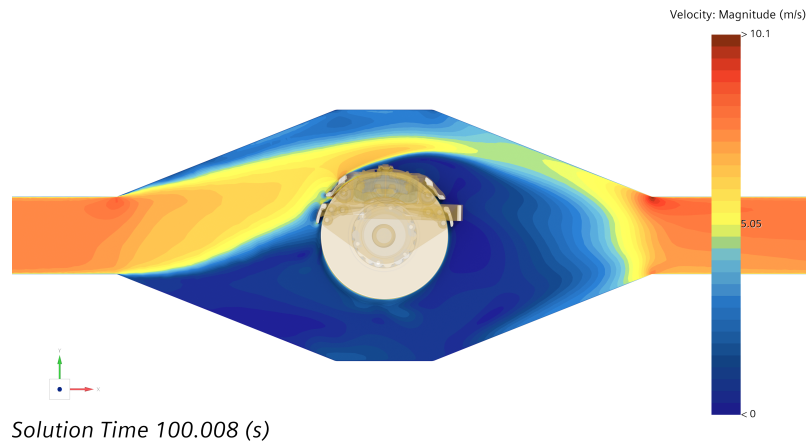


Figure 4.17: Air flow for the 40/60 braking cycle at 100 seconds - front view

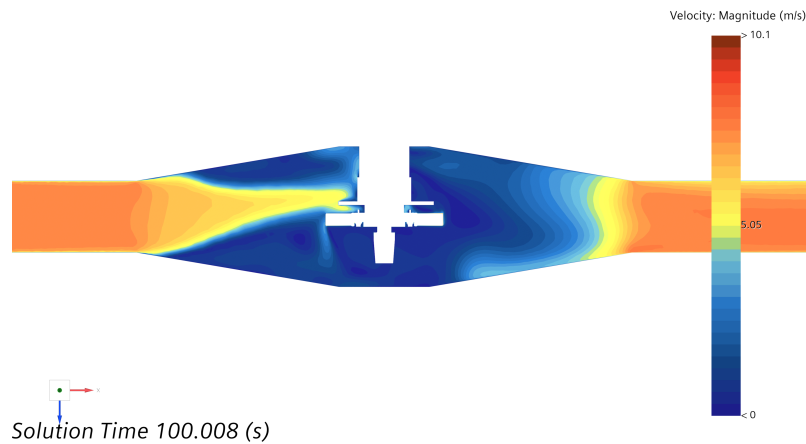


Figure 4.18: Air flow for the 40/60 braking cycle at 100 seconds - top view

The figures indicate that the inlet flow remains relatively concentrated and resembles a jet that does not spread uniformly across the full brake system. The majority of the flow does not affect the entire brake system, but rather affects it locally or flowing past it.

4.2.3.2 900/1800

For the 900/1800 braking cycle, the temperature distribution at physical times of 300, 600, 900, and 1200 seconds can be seen in Figure 4.19. In this figure the order of the time steps goes from top left down to bottom right. The figures show how the disc is heated up and how it has been cooled after its first 300 seconds of cooling.

4. Results

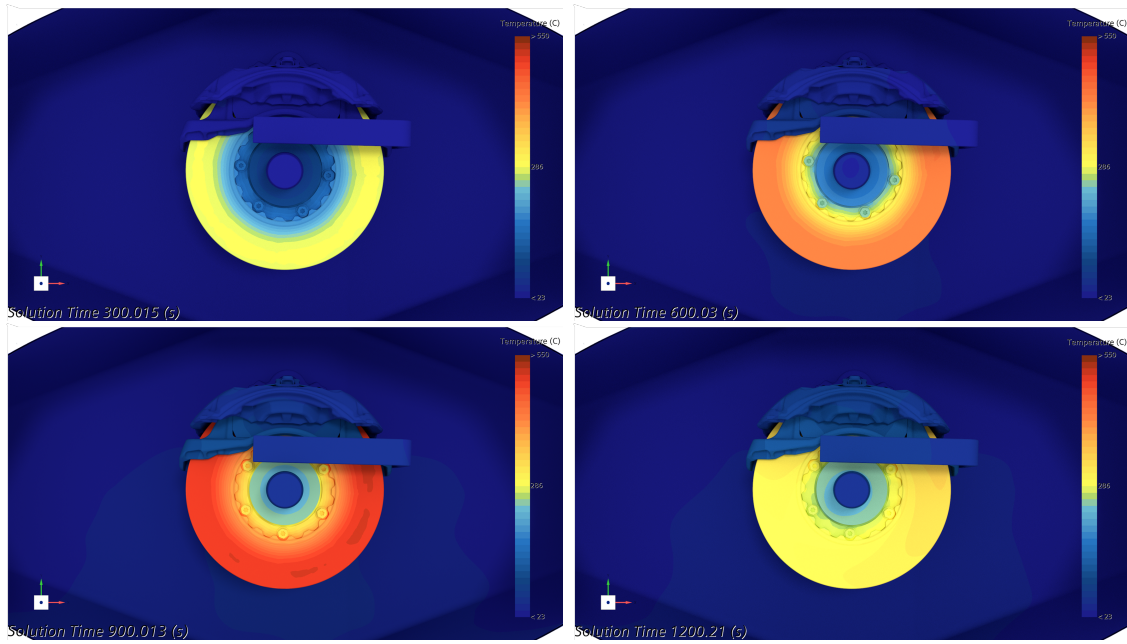


Figure 4.19: Surface temperature development over time from 300 to 1200 seconds

The flow evolution was also visualized in four captured scenes from 1150 to 1375 seconds in steps of 75 seconds, see Figure 4.20. The figure shows an oscillatory change in the steady-flow solution between the sampled time steps.

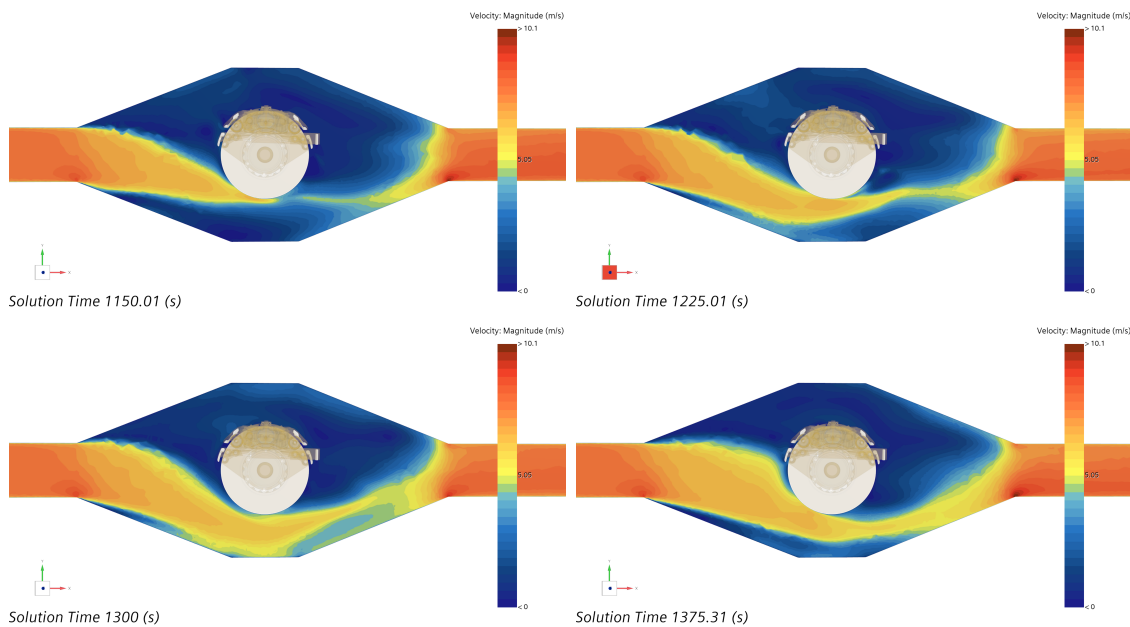


Figure 4.20: Air flow evolution over time during cooling stage of the 900/1800 braking cycle in the RBM setup

4.3 Convective Velocity Option

This section presents the comparison between the CVO setups and the RBM setup in terms of total solver CPU time, heat rates from the rotating parts, and maximum and average disc temperatures. The CVO setups use radiation in either the solid or the fluid domain.

4.3.1 40/60

First, the results for the 40/60 braking cycle are presented. In Figure 4.21, the temperatures are plotted. A clear difference can be observed in maximum temperature, with the CVO setups predicting lower values than RBM. For the average temperatures, they are similar between the setups during the braking phase. However, during the cooling phase, the CVO setups predict lower temperatures.

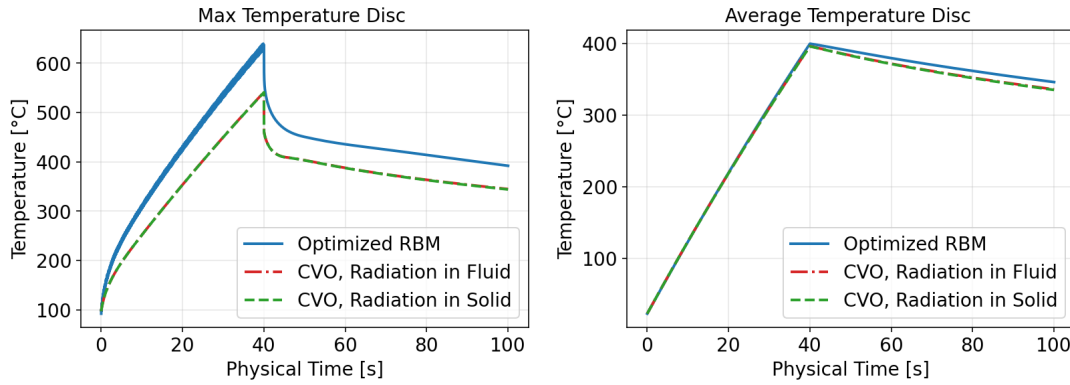


Figure 4.21: Maximum and average temperature in the disc with RBM and CVO for the 40/60 braking cycle

To compare the computational cost of the setups, the total solver CPU time is plotted for the three different configurations in Figure 4.22.

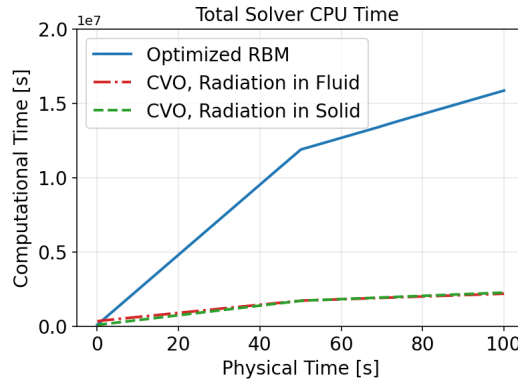


Figure 4.22: Total solver CPU time with RBM and CVO for the 40/60 braking cycle

Figure 4.22 shows that both CVO setups require less CPU time than the RBM setup. However, the CPU time is similar between using radiation in the fluid continuum and in the solid continuum.

The time saved and RMSE of the temperatures compared with the RBM setup can be seen in Table 4.5. From the table, it is possible to see that the two different CVO setups perform similarly to each other.

Table 4.5: The RMSE and time savings for the CVO setups compared to the RBM setup for the 40/60 braking cycle

	RBM	CVO Fluid	CVO Solid
Patch/Face Proportion	1	100	1
Nr of Beams	64	1024	64
Max Temp Disc after Braking [$^{\circ}C$]	634.24	541.11	540.72
Max Temp Disc after Cooling [$^{\circ}C$]	391.90	344.72	343.98
Avg Temp Disc after Braking [$^{\circ}C$]	399.32	395.88	395.61
Avg Temp Disc after Cooling [$^{\circ}C$]	345.80	335.66	334.87
Max Temp Disc RMSE	-	65.15	64.4
Avg Temp Disc RMSE	-	5.00	5.32
Time Saved [%]	-	86.2	85.8

The total, convective and radiative heat rates leaving the disc are plotted in Figure 4.23 which shows similar total heat rates for all three setups. The radiative heat rate is similar between the two CVO setups, which in turn have a lower magnitude than the RBM setup. For the convective heat rate, the CVO setups show a larger magnitude than the RBM setup. The CVO setup with radiation in the solid domain also shows a slightly higher convective heat rate magnitude than the CVO setup with radiation in the fluid domain.

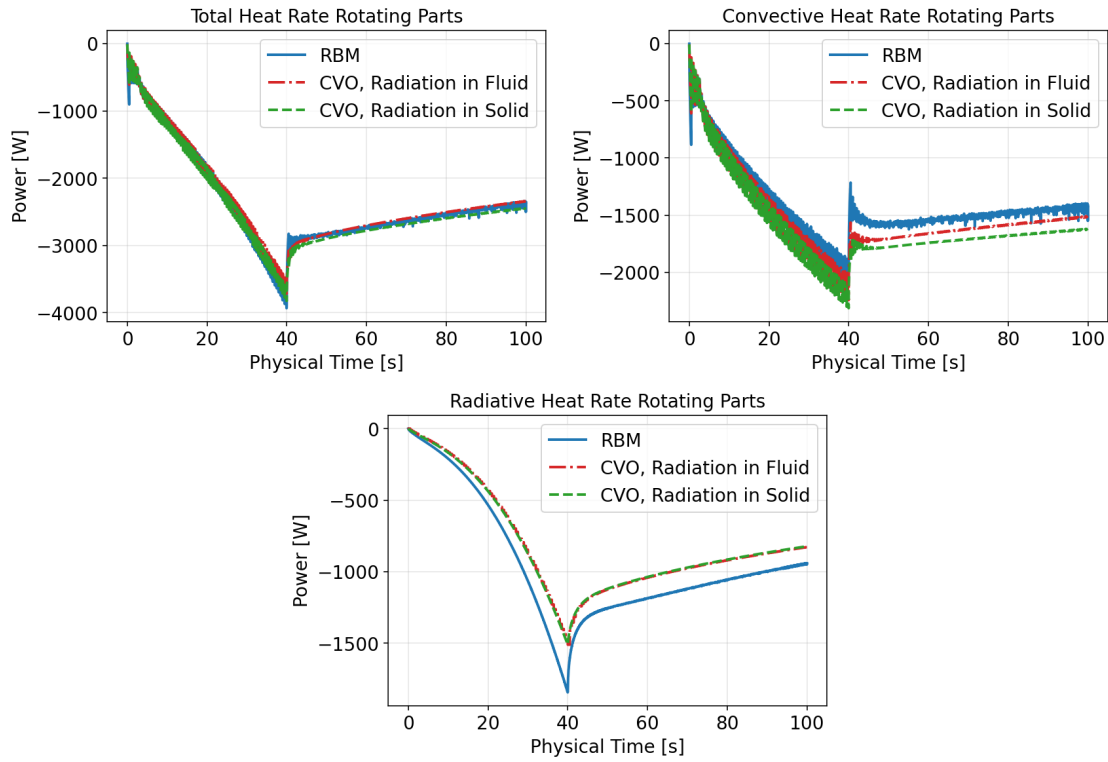


Figure 4.23: Total, convective and radiative heat rates from the rotating parts for the RBM and CVO setups for 40/60 braking cycle

4.3.2 900/1800

The temperatures are presented in Figure 4.24 where it is possible to see that the temperatures, particularly the maximum temperature, are lower for the CVO setups.

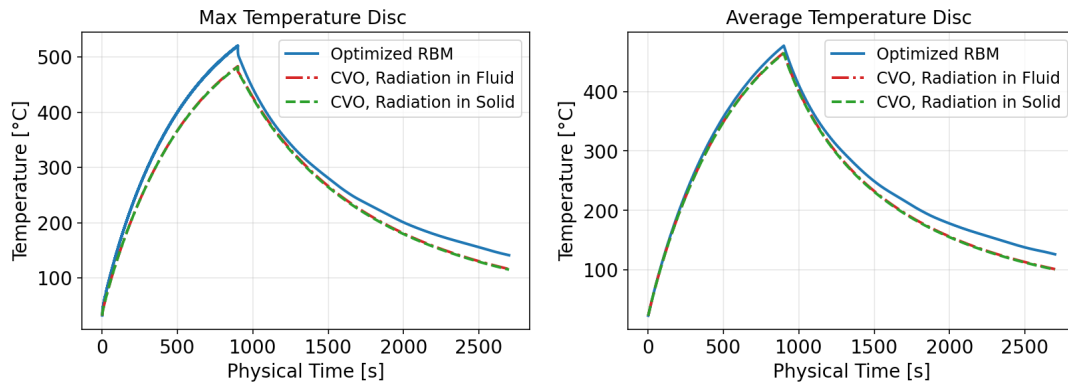


Figure 4.24: Maximum and average temperature in the disc with RBM and CVO for the 900/1800 braking cycle

In Figure 4.25, the total solver CPU time is presented for the three different setups. The figure illustrates that there is a large difference in total solver CPU time between the RBM and CVO setups. However, the difference between the CVO setups is small, where the setup with radiation in the fluid is faster.

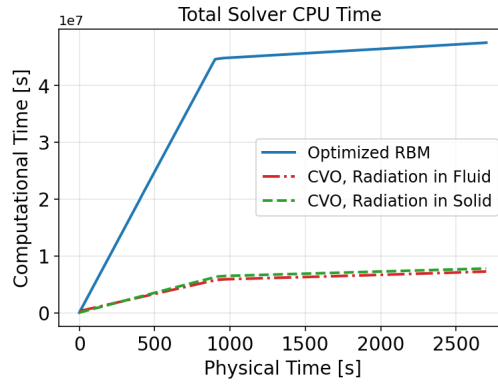


Figure 4.25: Total solver CPU time plot with RBM and CVO for the 900/1800 braking cycle

The RMSE and time saved with respect to the RBM setup is presented in Table 4.6. The table shows that the two CVO setups perform similarly relative to the RBM setup. It also shows that the CVO setups perform better in predicting the average temperature in the disc compared to the maximum temperature.

Table 4.6: The RMSE and time savings for the CVO setups compared to the RBM setup for the 900/1800 braking cycle

	RBM	CVO Fluid	CVO Solid
Patch/Face Proportion	1	100	1
Nr of Beams	64	1024	64
Max Temp Disc [$^{\circ}C$]	520.40	483.19	482.06
Max Temp Disc after Cooling [$^{\circ}C$]	141.00	115.86	114.95
Max Avg Temp Disc [$^{\circ}C$]	476.84	465.13	463.98
Temp Disc after Cooling [$^{\circ}C$]	126.15	101.21	100.43
Max Temp Disc RMSE	-	28.50	29.07
Avg Temp Disc RMSE	-	10.78	11.41
Time Saved [%]	-	84.6	83.5

In addition, the convective, radiative, and total heat rates from the disc are compared in Figure 4.26.

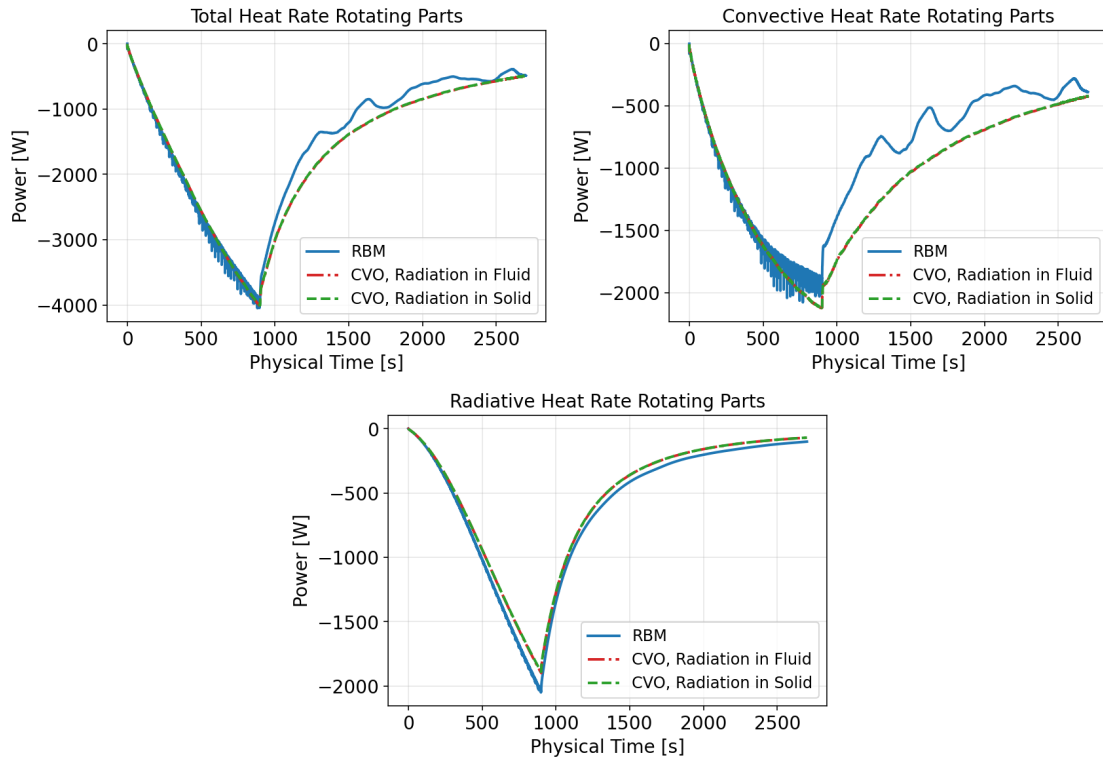


Figure 4.26: Total, convective and radiative heat rates from the rotating parts for the RBM and CVO setups for the 900/1800 braking cycle

From the figure above, it is possible to see that the two CVO setups give nearly identical results. The RBM setup has an oscillating convective heat rate in the cooling phase, as previously seen, and shows lower total and convective heat rates. The RBM setup, however, shows a higher radiative heat-flux magnitude.

4.3.3 Heat Application

This section visualizes the difference in surface-temperature distribution between the RBM and CVO setups. To show this, the surface temperature of the disc for different time steps are presented for the RBM setup in Figure 4.27 and for the CVO setup in Figure 4.28.

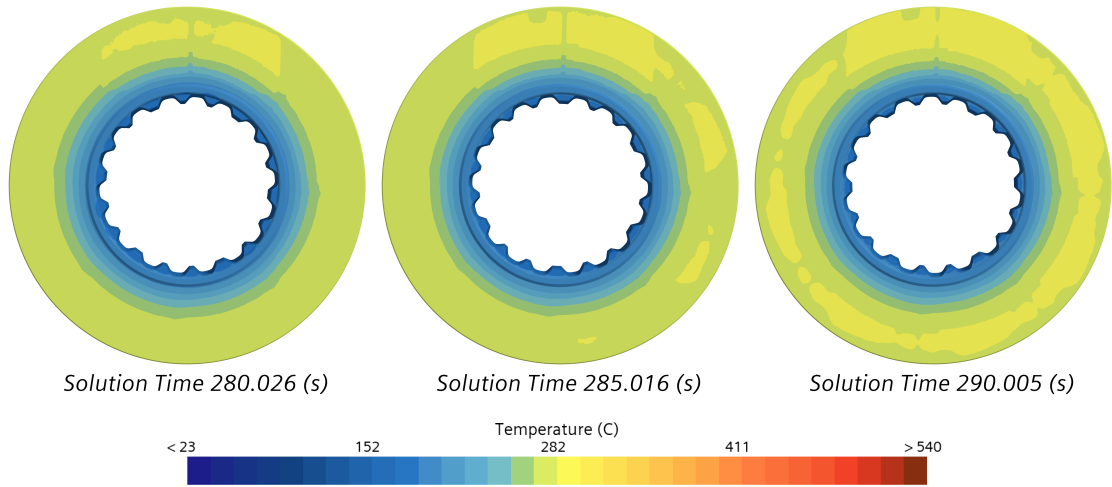


Figure 4.27: Heat generated for the RBM setup at the disc surface

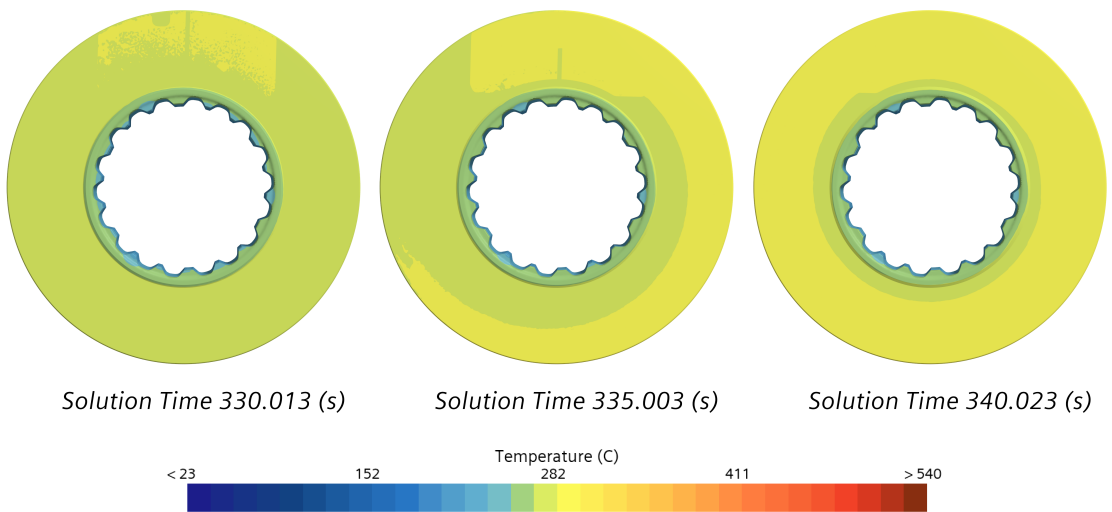


Figure 4.28: Heat generated for the CVO setup at the disc surface

It can be seen that for the RBM setup, the applied heat results in a pattern with a clear temperature gradient towards the center of the disc which remains cooler. In the CVO setup, however, the temperature has smoother gradients and distributes the heat more evenly. The same surface-temperature level is also reached earlier in the RBM setup.

4.3.4 Temperature

To further compare the surface temperature distributions between the setups, the temperature field was captured for the brake system at two different time steps. The first time step is at 900 seconds where the temperature reaches its maximum value,

see Figure 4.29. The second time step is at 1200 seconds where the brake has been cooling for 300 seconds, see Figure 4.30.

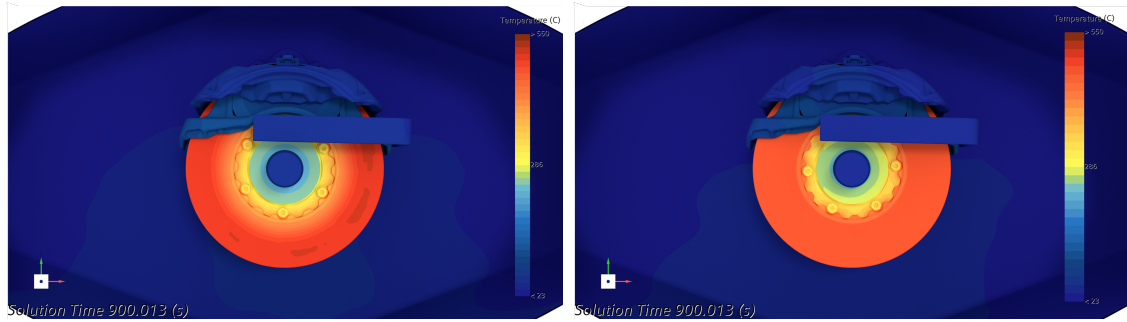


Figure 4.29: Surface temperature comparison between the RBM (left figure) and CVO (right figure) setups at 900 seconds

At 900 seconds, the RBM setup shows a higher surface temperature on the disc, while the central region remains cooler than in the CVO setup.

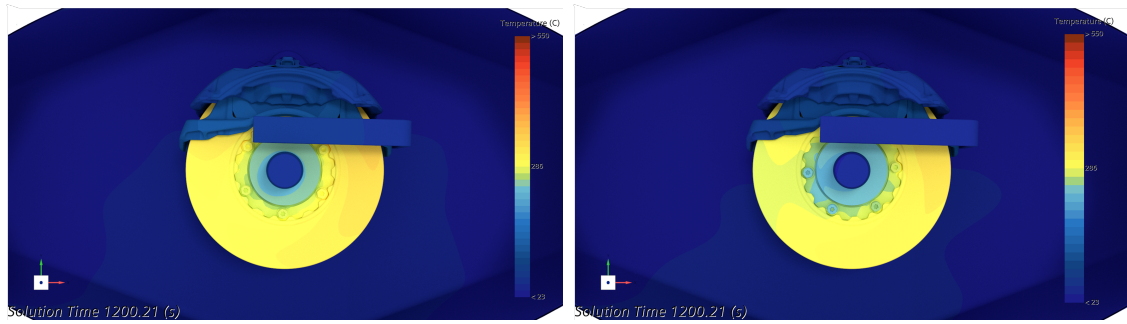


Figure 4.30: Surface temperature comparison between the RBM (left figure) and CVO (right figure) setups at 1200 seconds

From the second comparison, at 1200 seconds, it can be seen that the surface temperature in the CVO setup is lower than that of the RBM setup. Furthermore, after stationary cooling, the left side of the disc is colder than the right side for both setups.

4.4 Verification of Simulations

To verify the simulations, the temperatures at the thermal probes are compared in Figure 4.31. For the CVO method, the setup with radiation in the solid domain was used, but either setup would have been suitable because both performed similarly. The braking cycle used for comparison is the Rig braking cycle. For the values from the rig, the values are averaged within each pair of temperature probes except for T5, since there is no T6. The probes in the simulations are located to the left of the center of the disc which, as shown in Figure 4.30, corresponds to the coldest part of the disc during stationary cooling.

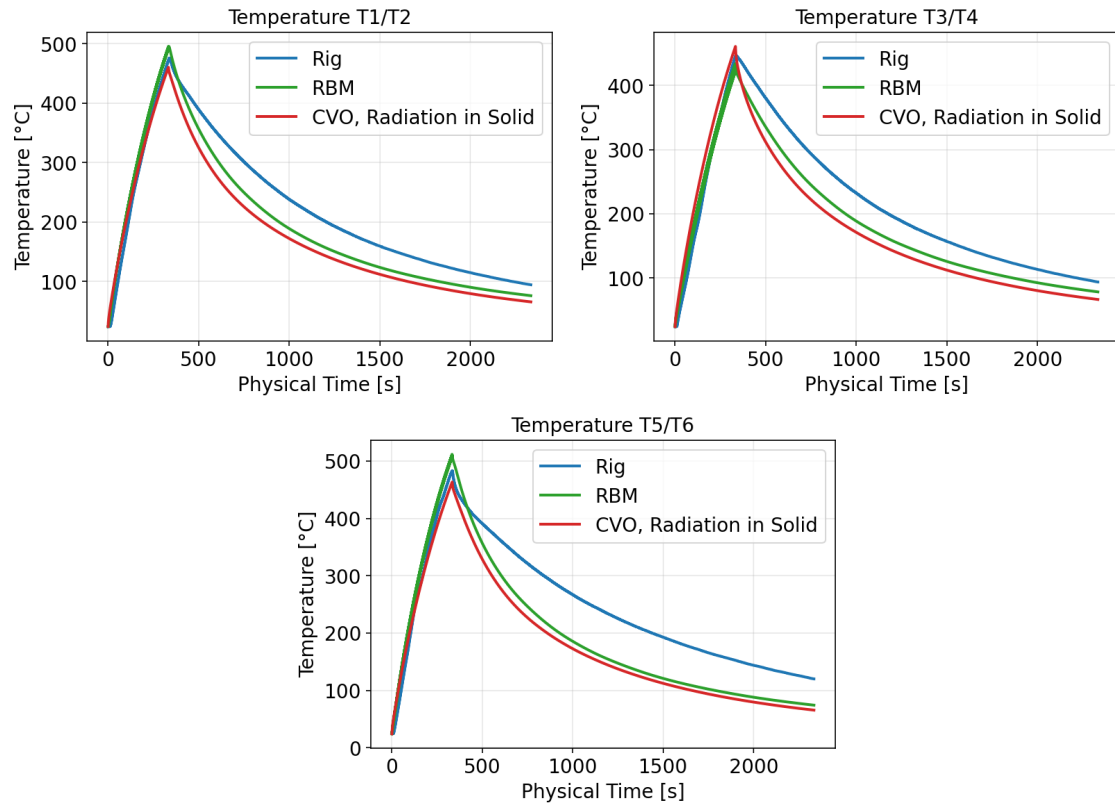


Figure 4.31: Comparison of thermal probes in the rig, T1-T5, and the point probes in the simulations

The figure shows that, during the braking phase, the temperature evolution is similar in the rig and the simulations. However during the cooling phase, the probes in the simulation have lower temperatures than the averaged probe pairs in the rig. The temperature differences at the end of the braking phase and after cooling are summarized in Table 4.7.

Table 4.7: The temperature after braking and cooling for the simulations compared to the rig

	Rig	ΔT RBM	ΔT CVO Solid
T1/T2 after Braking [$^{\circ}C$]	475.86	19.57	-15.36
T1/T2 after Cooling [$^{\circ}C$]	94.43	-18.38	-28.80
T3/T4 after Braking [$^{\circ}C$]	447.03	-12.81	13.95
T3/T4 after Cooling [$^{\circ}C$]	93.64	-15.43	-27.25
T5/T6 after Braking [$^{\circ}C$]	483.30	28.64	-19.66
T5/T6 after Cooling [$^{\circ}C$]	120.05	-45.83	-54.5

The temperatures measured by all thermal probes in the disc are plotted in Figure 4.32. The figure shows that, for both probe pairs T1/T2 and T3/T4, one probe records a higher temperature than the other. At the final time point, the difference between T1 and T2 is $54.84^{\circ}C$. For the simulated RBM setup at the same time point, the maximum temperature difference in the entire disc is $22.09^{\circ}C$.

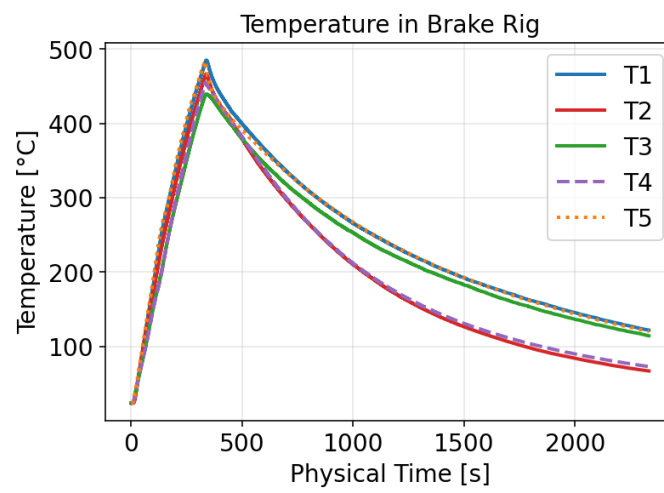


Figure 4.32: All measured individual temperatures measured in the brake rig, T1-T5

5

Discussions

This chapter discusses the findings of the study, with focus on how choices related to RBM, CVO, and radiation settings affect temperatures and computational time. It also addresses limitations of the present approach and possible improvements.

5.1 Temperature Monitors

Using the maximum disc temperature as the main comparison metric could lead to misleading comparisons. When monitoring the maximum temperature, the cell from which the value is extracted may exhibit a temperature peak due to, for example, a cell volume being close to zero. This would mean that the comparison is not representative of the real temperature. When simulating the 40/60 braking cycle, a maximum difference between the maximum and average temperature can be seen to be 235°C, see Figure 4.21 and Table 4.5. However, when looking at the difference for the 900/1800 braking cycle the difference is at around 44°C, see Figure 4.24 and Table 4.6. The larger difference in the 40/60 braking cycle is expected due to it heating up faster and therefore, getting a higher surface temperature while not having enough time to conduct the heat away. Still, the plots could show non-representative data.

Instead of using a maximum-temperature plot, while still capturing surface-temperature data, an average surface temperature would be a better way of capturing the temperature. In this kind of plot the same comparisons could be made, such as how the surface temperature differs between the setups but with a higher certainty of representation.

A similar issue applies to the use of point probes in the simulations. During rotation, fixed point probes effectively sample a circumferential temperature distribution. However, once the disc stops rotating, the probe values become sensitive to the final angular position of the disc. A more robust representation of the rig probes would therefore be to use a line probe around the disc circumference at the corresponding radial and axial location, and monitor its averaged value.

5.2 Radiation

How radiation affects the simulation is discussed in this section. It focuses on differences between implementing and not implementing radiation and how the setups

can be optimized.

5.2.1 Radiation Modeling Approach

For Setup A that uses RBM and the 40/60 braking cycle, the radiation model was implemented in the fluid domain which means that the radiation was calculated every time the fluid domain is solved. Figure 4.1 shows that the temperature is nearly identical between the simulations with and without radiation. It is also possible to see that the radiative heat rate does not get added to the total heat rate in Figure 4.2. Together, these observations indicate that the radiative heat rate is not accounted for in the energy balance of the simulation with Setup A. For the simulation when the disc was stopped during the cooling phase, it is possible to see that the radiative heat rate gets added to the total heat rate in the cooling phase, see Figure 4.4. Why radiation in the fluid domain does not function correctly in combination with RBM in *Star-CCM+* is unclear. It was initially suspected that this might be related to the steady treatment of the fluid domain, and thereby of the radiation model. However, due to the radiation working as expected when the RBM is stopped, this is not the case.

When looking at the temperatures for Setup B in Figure 4.5 for the 40/60 braking cycle using RBM, it is possible to see that there is a small temperature difference between the cases with and without radiation. Also, when comparing the heat rates for Setup B in Figure 4.6 with Setup A in Figure 4.2 it is possible to see that the radiative heat rate is of similar magnitude for the two setups. However, for Setup B, the total heat rate is equal to the sum of the radiative and convective heat rates. The observed temperature difference and the consistent heat rate balance indicate that the radiative contribution is accounted for in the energy balance.

5.2.1.1 Time Savings

There is a large difference in total solver CPU time between the radiation implementations in Setup A and Setup B with roughly $4 \cdot 10^6$ and $8 \cdot 10^6$ seconds, respectively, see Figure 4.3 and 4.7. Since A uses radiation in the fluid domain, the radiation is calculated every time the fluid domain is solved which is every 0.5 seconds. For Setup B, the radiation is implemented in the solid domain and therefore, calculated every time step due to the domain using an unsteady timescale. For comparison, in Setup A during a 40/60 braking cycle, the radiation was calculated once every 38 time steps. This means that radiation is solved much more frequently in Setup B than in Setup A, which is most likely why Setup B requires roughly twice as much CPU time.

5.2.2 Thermal Effects of Radiation

For the 900/1800 braking cycle using Setup B, it is possible to see that there is a significant temperature difference between the simulations with and without radiation. The difference becomes visible during the heating phase at around 350 °C. It keeps growing throughout the braking phase reaching around 83 °C for maximum

temperature and around 74 °C for average temperature in the disc. Why the difference in temperature gets larger at the end of the braking phase can be explained by Figure 4.9. In that figure it is possible to see that the radiative heat rate reaches its maximum magnitude at the end of the braking phase where the temperatures are the highest. This is consistent with the Stefan-Boltzmann law in Equation 2.20, due to its fourth-power temperature dependence. The higher radiative heat rate results in more cooling of the brake and therefore lower temperatures.

However, there is no significant difference in temperature for the 40/60 cycle using Setup B, see Figure 4.5. When looking at the radiative and convective heat rates in Figure 4.6, it is possible to see that they are of similar magnitude. Furthermore, the radiative and convective heat rates also have similar magnitude as for the 900/1800 braking cycle, see Figure 4.9. At first glance, this could suggest that a similarly significant temperature difference should also appear in the 40/60 braking cycle. However, since the 40/60 braking cycle is performed during a shorter period of time, 100 seconds compared to 2700 seconds, that is not the case.

The thermal energy leaving the rotating parts region from both radiation and convection is compared to the total thermal energy applied to the disc. In the 900/1800 braking cycle, the radiation is affecting the temperature for a longer time and by that, the thermal energy due to radiation and convection has a larger impact. For the 900/1800 braking cycle, the thermal energy lost through radiation from the rotating parts is 22.6% of the energy applied, see Table 4.2, but for the 40/60 it is just 3.2%, see Table 4.1. The rest of the thermal energy applied is leaving the domain through conduction or saved as internal energy.

5.2.2.1 Time Savings

It is possible to see that the total solver CPU time can be drastically decreased when radiation is not accounted for using Setup B, see Figure 4.7 and 4.10. For both braking cycles, the computational time could be decreased by around 70%. This reduction is larger than that for Setup A, see Figure 4.3 with only around 20%. This is again due to the fact that in Setup A, the radiation is implemented in the fluid domain, which is solved less often than the solid domain.

5.2.3 Optimized Radiation

Given the increased computational time of Setup B, it is more important to find optimized radiation settings to retain reasonable computational time. When comparing Simulations 1-6 in Table 4.3 and 4.4, it is possible to see that they all have a small RMSE below 0.5. Simulations 3 and 4 required the least CPU time while also giving the lowest RMSE values, and were therefore considered the most promising options. Both of them have the same *Number of Beams*, 64 but different *Patch/Face Proportion*, 2.5 and 1, respectively. Simulation 4 was chosen as the best setup but both have their strengths. Simulation 3 has the lowest RMSE and could therefore be seen as the best tested option but Simulation 4 should, in theory, require the least amount of CPU time since it reduces the amount of patches used.

Furthermore, it should be stated that in theory, the higher the values of the two parameters tested, the higher the accuracy. This was not the case in these simulations where Simulation 3 and 4 was closer to Simulation 1 than Simulation 2. It is therefore hypothesized that the observed differences are mainly caused by numerical variations in the patch and view-factor calculations, and that all tested configurations perform similarly in terms of accuracy.

When it comes to computational time, Simulation 3 and 4 performed identically which was not expected. The two configurations were checked thoroughly so that the parameters were not the same in the two simulations which they were not. A likely explanation is variation in communication performance between the computed nodes used for the different simulations. Even though all simulations used adjacent nodes from the node number, it does not mean that they are physically neighboring each other and can therefore, sometimes result in a higher solver CPU time. Restricting the comparison to simulations run on adjacent node numbers was still considered the fairest approach, although it does not remove all uncertainty.

It should be noted that the study only covers a limited number of parameter combinations. To be able to get an optimal simulation setup, a more thorough investigation would have to be performed. It would require a larger variation of *Number of Beams* and *Patch/Face Proportion*. Different braking cycles would also be beneficial.

5.2.3.1 Parameters

When looking at the effects of reducing *Number of Beams* and *Patch/Face Proportion*, it is not possible to see any clear pattern of which parameter has the largest impact on CPU time and accuracy. It is possible, however, to see that reducing both parameters simultaneously resulted in a lower computational time without drastically reducing the accuracy. A reason for why the accuracy is not seriously affected could be due to the fact that the reference simulation, Simulation 1, uses a *Number of Beams* and *Patch/Face Proportion* of 256 and 10 respectively. Those values are lower than the default values of 1024 and 100 which makes a difference in accuracy possible between the default settings and the settings for Simulation 1.

Rather than reducing both parameters at the same rate, it is likely that *Number of Beams* and *Patch/Face Proportion* affect the solution differently depending on the geometry. For a complex geometry with different angles, it could be important to use a higher *Number of Beams* to maintain accuracy since a higher value captures more directions. A high value of *Patch/Face Proportion* increases spatial resolution and could therefore be important if hot spots and sharp temperature differences have to be resolved properly.

5.2.3.2 Time Savings

When comparing the computational time of the different configurations, seen in Figure 4.12 and Table 4.3, it can be seen that up to 14.8% of computational time

can be saved compared to Simulation 1. This time saving would probably be larger if a comparison would be made with a simulation using default settings. Simulating braking cycles with default settings was the plan in the beginning of the project but due to limited time, this plan was modified to obtain all results within the project period.

5.3 Heat Transfer

This section discusses heat transfer considerations and focuses on energy coupling and convection.

5.3.1 Choice of Energy Coupling

The argument for an explicit energy coupling in this case is that it would ensure that the rotating parts would maintain a 100% interface between the fluid and the solids over time. When initializing the solution, the interface is set once and it keeps that connection through the RBM. The problem with this is that the part of the disc that is under the pad never interacts with the air and therefore, does not lose heat through convection or radiation. This will result in a higher temperature in that section while the rest of the disc will always experience convection and radiation, even when a section is under the pad. This would result in a slightly lower temperature than what is expected to be physically accurate there. One small advantage of using explicit energy coupling is that when heat is not applied, there is no need to rotate the disc because the only thing that needs to be spread out in such a setup is the applied heat. Without rotation the computational cost would decrease. This is, however, just a result of the loss of physics of cooling.

Implicit energy coupling will automatically update the interface for every time step when the disc is rotating. This ensures an accurate heat flux between the air and the solids. However, in parts that are not axisymmetric, the interface could be lost in combination with RBM due to the boundaries not aligning. This can most simply be visualized by the bolts not lining up with a bolt's pocket in the air domain. This would reduce the convective heat flux in those regions, resulting in slightly higher local temperatures. The choice is not completely simple and it is reasonable to assume that there is a limit where the captured amount of non-axisymmetric boundaries becomes more important than the evenly distributed cooling of the disc.

5.3.2 Fluctuating Convection

When analyzing all heat rates from convection for both the 900/1800 and 40/60 braking cycles, the convection develops as a jagged line over time, as seen, for example, in Figure 4.6. This was thought to be because of two main reasons. The first is because of how the flow is solved. Since convection depends on the temperature difference across the boundary, and the flow field is only updated every 0.5 seconds, some fluctuation in the convective heat rate is expected. The reason for this is that while solving the solid domain, the disc surface is heated up while the air stays at

a constant value, increasing the difference in temperature which increases the convective heat rate. But after some time steps in the solid domain, the air domain is solved again which heats it up and thereafter decreases the difference in temperature.

However, when analyzing the 900/1800 braking cycle results, it can be seen that when the disc has stopped, the jagged behavior disappears, see Figure 4.9. For the 40/60 cycle, the jagged lines are left in the cooling phase where it is still rotating, see Figure 4.6. This suggests that the jagged behavior mainly originates from the RBM treatment of the disc. Therefore, they are thought to be an effect of the interface capturing proportion going up and down with the implicit energy coupling. Due to the two coupling methods having their own drawbacks, the safest but more computationally expensive solution would be to incorporate an axisymmetric fluid part around the non-axisymmetric parts that would solve the fluid in an unsteady time scale while rotating together with the disc. The primary choice for this shape would be to enclose the whole disc but this is not possible when the pads are applied directly on the surface of the disc. A second option would be to enclose only the center of the disc which in this geometry is where the interface is lost.

A third solution to the interface problem is to use data mappers instead of interfaces. This would assign to each boundary cell the value of the nearest cell on the other domain, thereby ensuring full coverage of the interface but potentially introducing local interpolation errors. Another key advantage of data mappers is that it enables a manual control over how often to solve the coupling between the domains and therefore, also how often to compute radiation [34].

5.3.3 Oscillating Convection

In addition to the fluctuating convective heat rate from the RBM, an oscillating heat rate can be seen in the same plot, Figure 4.9, after the rotation has been stopped. This is most likely an effect of the flow changing over time which can be seen in Figure 4.20. The air flow is moving around in an oscillating manner, resulting in the flow velocity near the disc sometimes being high and sometimes low. When the flow stagnates near the disc, convective heat transfer decreases because the HTC is reduced since the surrounding air temperature increases when hot air is not efficiently removed.

5.3.4 Heat Application

The heat is applied in a combined interface of both sides of the disc and, in this interface, the heat is applied constantly. Both sides have the same amount of heat application and also, the same on every cell of the surfaces. Applying the same heat flux to all cell surfaces is a practical simplification, but in real braking scenarios the heat generation differ between the two sides of the disc and over the braking surfaces.

Another aspect of these setups is what proportion of the heat goes into the disc

compared to the pad. This is mostly an effect of effusivity but also an effect of thermal contact resistance which was set to zero for these simulations. Effusivity should however, be captured quite well due to the temperature dependent material properties of the disc and pad. The thermal contact resistance does not affect comparisons between different simulations but does affect comparisons with experimental data.

5.4 Convective Velocity Option

In this section, it is discussed how the CVO setups perform for the different braking cycles.

5.4.1 Temperature

In general, the CVO setup appears to perform reasonably well when compared with the RBM setup, see Figure 4.21 and 4.24. Especially in the plots showing the average temperature of the disc with a difference of just 5-10°C for the 40/60 braking cycle and 12-25°C for the 900/1800, according to Tables 4.5 and 4.6. It can be seen, however, that the maximum temperature plots differ more. For the maximum temperature plots, the difference in temperature is 47-93°C for the 40/60 braking cycle and 36-37°C for the 900/1800, following the same tables as before. The CVO setups distribute the heat more evenly, whereas RBM produces higher local surface temperatures that are not redistributed as quickly through the disc, resulting in stronger temperature peaks. This higher surface temperature can also be seen in Figure 4.27 and 4.28 where the same surface temperature is reached earlier for the RBM setup compared to the CVO setup.

Why the 40/60 braking cycle has higher peaks in the RBM setup compared to the CVO is assumed to be due to the higher torque and rotational speed and therefore higher generated heat. In that braking cycle, the setups have a small time to distribute the heat in the disc, affecting the RBM setup more due to it not distributing it as fast.

5.4.2 Heat Rates

When inspecting the heat rates out of the rotating parts, there is generally more convective heat transfer in the CVO setup but more radiative in the RBM. The heat rates can be seen in Figure 4.23 and 4.26. Why there is more convection in the CVO setup is probably due to the interface being fully captured compared to the RBM setup where the implicit energy coupling resulting in a loss of interface area. It may also be influenced by the unsteady flow behavior observed in the RBM setup. The reason for more radiative heat rate over the boundary in the RBM setup is explained by the surface of the disc reaching higher temperatures in that setup, as previously discussed. A higher surface temperature makes the disc radiate more energy.

In Figure 4.23 showing the 40/60 braking cycle it can be seen that the implementation of radiation in the solids combined with CVO gives higher convective heat rate than the setup using radiation in the fluid. However, when comparing the convection in Figure 4.26, showing the 900/1800 braking cycle, the heat rate is identical for the two setups. The second result was more expected than the first because the only difference between the setups is in which continuum radiation is enabled which should not affect the convection. However, a difference could arise if differences in radiative heat transfer lead to different disc temperatures in the two setups, which would in turn affect convection. However, when looking at the radiative heat rate in the same figure, the two different configurations have very similar radiative heat rates. Since the two configurations give nearly identical results for the 900/1800 cycle and very similar radiative heat rates for the 40/60 cycle, the difference in convective heat rates for the short cycle may be due to software-related behavior, although this remains uncertain.

5.4.3 Difference in Flow

When simulating the 900/1800 braking cycle, the flow behaves differently for the CVO and RBM setups. The difference can be seen when the cooling stage is active as visualized in Figure 4.20. In the whole CVO simulation and in the braking stage of the RBM simulation, the flow is stable and changes little over time. Why the flow starts to oscillate for only the RBM setup is unclear because the fluid domain should be the same for the two setups. In the fluid domain, a TVS is set at the boundary of the rotating parts that simulates the rotation for both setups which is disabled during the cooling stage. The only expected differences are the temperature of the brake surfaces that could affect the flow slightly and the generated mesh. Although the same mesh settings were used, the generated meshes are not identical, which may influence the flow solution. The oscillations are the same for all different radiation setups and it is therefore hypothesized to not be an effect of any numerical error. Even though the reason why it oscillates in the RBM setup but not in the CVO is unclear, the flow can be seen as unstable during this braking cycle.

5.4.4 Heat Application

The main difference between the setups is how the heat is applied and spread over the disc, see Figure 4.27 and 4.28. For the RBM setup, a pattern appears in which an outer ring reaches a higher temperature. This is probably due to the caliper holder and axle in the middle of the disc being a big heat sink and taking heat from the inner side of the pad. The outer side does not have a cold heat sink close by and does not lose heat as fast. The heat flux through the disc to the caliper holder and axle can be seen by how the temperature is radially decreasing towards the middle of the disc.

In the CVO setup, the generated heat is smoother and this is assumed to come from the numerical diffusion smearing the gradients of the second term in Equation 2.15. This is hypothesized to be a result of using a mesh that is not directed, follow-

ing the rotation of the disc. The diffusion is applied over all cell surfaces, even when they are not aligned, which sends heat radially, smoothening the temperature field. Based on these observations, RBM appears to provide the more physically realistic heat distribution, at least when the CVO setup is used without a directed mesh.

5.4.5 Time Savings

From Table 4.5 and 4.6 it is possible to see that the two CVO setups save around 86% of computational time for the 40/60 braking cycle and 84% for the 900/1800 braking cycle. These values are for the CVO setup with radiation in the fluid domain due to it being slightly faster than the setup with radiation in the solid domain. The RBM simulation for the 900/1800 braking cycle took around 50 hours to execute and if that number could be reduced by 84% that would mean a calculation time of around eight hours instead. That would make it possible to submit a simulation at the end of a working day, and obtain the results in the following morning which is a clear advantage. Therefore, CVO can be worth using, especially for longer and less extreme braking cycles. However, the reduced computational time comes at the cost of lower accuracy than RBM, and the choice between the two approaches should therefore depend on whether computational speed or predictive accuracy is prioritized.

5.5 Experimental Rig

In this section, the verification of the simulations with the experimental rig will be discussed. The differences discussed are mainly the temperature difference between different probes in the rig compared to the simulation and the geometry with its different materials.

5.5.1 Temperature Measurements

When studying the results from the simulations and comparing it with the data from the rig in Figure 4.31 and Table 4.7, the temperature behaves similarly in the braking phase. As previously discussed, in the braking phase, the singular thermal probe is sufficient but in the stationary cooling phase, the singular point in which the probe is located could affect the comparison validity. This point corresponds to an area of the disc subjected to more cooling which results in lower temperatures. Why the temperature is lower for the simulations in the cooling phase could be partly explained by this.

To further analyze this, the simulated temperatures corresponding to T1/T2 and T3/T4 are compared separately with T1-T4 from the rig in Figure 5.1. In the figure, it can be seen that T2 and T4 are close to the simulations while T1 and T3 are substantially hotter, which supports the view that the probe representation in the simulations is not optimal.

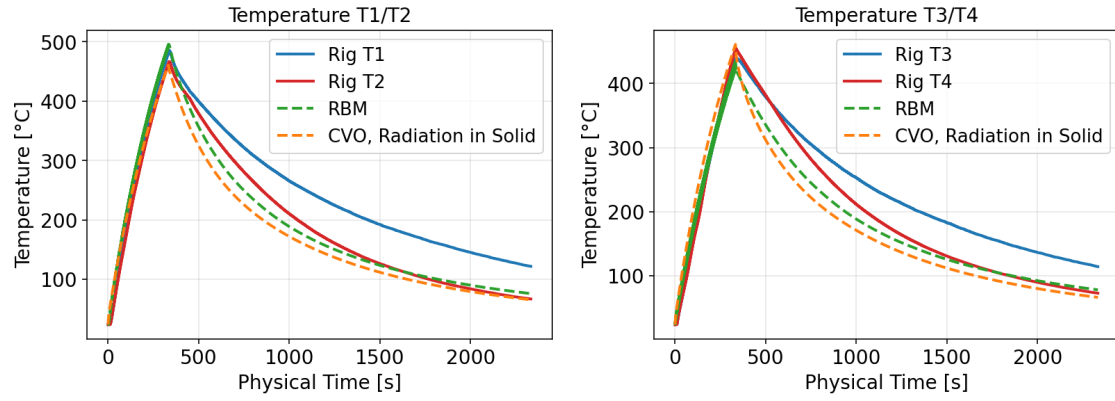


Figure 5.1: Comparison between rig and simulations with just T1/T2 and T3/T4

In the experimental rig, the disc rotation is also stopped but the orientation is not monitored. This means that the two probes could be fixed in any direction and one probe could be located in a cold region and the other in a hot region. It is expected to be hottest under the caliper, including the area under the pads, and it is expected to be coldest on the left side of the disc due to the forced convection. However, the difference between T1 and T2 in the rig is 54.84°C at the end of the cooling phase while the maximum temperature difference in the disc at the same time step is only 22.09°C . This suggests that the discrepancy between the rig and the simulations cannot be explained solely by the probe placement.

5.5.2 Geometry and Materials

One difference between the rig and the simulations is that the geometries are not identical. Some components inside the caliper were not included in the simulations and were instead represented as void regions. This void makes the brake system lose some thermal mass. In principle, this reduction in thermal mass would be expected to increase the simulated temperatures, which is not what was observed. Some other parts that are missing are the piston and cylinder which also makes the brake system lose some thermal mass. These two were however, expected to not affect the temperature much because they are far from the disc and pads and therefore, not getting heated up substantially.

There were also differences which could affect the flow more than the temperature. These were mostly cables connected to the thermal probes and the black plate behind the caliper which were not modeled identically in the simulation geometry.

Furthermore, the material properties of the geometry are not exactly correct. While the pad material properties and other parts of the disc are known to be correct, all rig-related materials were modeled as a standard steel material which also got applied on all smaller metallic parts in the brake, such as the bolts. All plastic parts were also modeled as a standard plastic material.

6

Conclusions

This chapter summarizes the main conclusions regarding radiation modeling, energy coupling, and the use of CVO. It also evaluates how well the final simulation setups agree with the experimental rig and outlines relevant future work.

6.1 Radiation

Radiation should be included in these simulations, especially for the 900/1800 braking cycle, where neglecting radiation leads to significant over prediction of brake temperatures. This is due to the strong temperature dependence of radiative heat transfer at the high brake temperatures reached in the simulations. When used with RBM, excluding radiation can reduce computational time by 70%.

For RBM, radiation had to be implemented in the solid domain in order to ensure energy conservation. This doubles the computational cost due to patches and view factors having to be calculated every time step. This is computationally expensive compared with radiation implemented in the fluid domain, where the view factors are calculated every time the steady-state fluid domain is solved.

However, to decrease the computational cost, the *Number of Beams* and *Patch/Face Proportion* can be decreased drastically while still producing accurate results compared with setups using higher parameter values. When comparing the computational time from Simulation 1 to 4, the computational cost decreases by 14.8% which is considered a substantial performance increase. No clear dominance of either parameter on computational time could be identified within the tested range. However, the combined positive influence of these parameters on view factor calculation is realized.

6.2 Energy Coupling

The choice of energy coupling affects the results in the setups using RBM. With implicit energy coupling, parts of the interface are lost over time, which contributes to fluctuations in the predicted convective cooling of the disc. However, the implicit coupling is deemed best for this setup but with increasing complexity of the geometry, like in the case of ventilated discs, explicit coupling could result in better accuracy.

To solve the loss of interface, either data mapping or a rotating region of air that is solved in an unsteady time scale should be implemented. Data mapping solves the interface capture by always mapping values to the nearest neighbor. A rotating fluid region can preserve a complete interface, but at the cost of increased computational effort due to transient flow simulation.

6.3 Flow Solver

For the 900/1800 braking cycle with stationary disc cooling, the flow results from the RBM setup indicate unsteady behavior. This suggests that a purely steady RANS treatment is not fully appropriate for that part of the cycle. In the spinning phases, the flow is steadier so either another flow solver should be used or the disc should keep spinning in all simulated braking cycles. Why the unsteady behavior appears in the RBM setup and not in the CVO setup is unclear.

6.4 Convective Velocity Option

CVO gives average disc temperatures that are reasonably close to those from RBM, but predicts lower maximum temperatures due to numerical diffusion. This diffusion reduces local surface temperatures and thereby decreases the radiative heat transfer compared with RBM. However, in the CVO setups, the interface is fully intact which gives a higher convective heat rate compared with the RBM setup. The diffusion gives also hotter temperatures close to the axle and caliper holder which causes more heat to be lost there through conduction. Implementing a directed mesh that follows the rotation of the disc is hypothesized to decrease the smearing due to numerical diffusion.

Overall, CVO is a useful alternative when computational time is a limiting factor, as it reduces computational time by about 84 to 86% compared with RBM. This would especially be of interest in pre-studies where maximum accuracy is not the highest priority.

For the CVO setups, enabling radiation in either the solid or fluid domain had little effect on the results or computational cost.

6.5 Verification with Rig

Comparison with the rig shows that the RBM and CVO setups perform similarly in terms of overall accuracy. It can also be concluded that both setups perform well in the braking phase but vary in performance in the cooling phase depending on which thermal probe is considered.

Despite reasonable agreement during braking, both simulations under-predict the temperature during the cooling phase. The magnitude of the real under-prediction

is difficult to conclude due to not knowing the stationary orientation of the probes in the rig. However, because the probes in the simulations are located in the coldest region, a smaller temperature difference than presented in the results is to be expected. It can still be concluded that the unknown orientation can not increase the temperature of the simulations enough to match the data, implying that the simulations would still under-predict the temperatures.

6.6 Future Work

The highest priority for future work is to capture the incomplete interface in the present coupling approach. Therefore, implementing data mappers or a rotating fluid region is considered important to investigate.

A directed mesh for the CVO setup is also of interest, since it is expected to reduce numerical diffusion and improve agreement with RBM.

A shorter analysis that could also be of interest is to compare the parameter settings of radiation to the default values. This would show how much the optimized setup actually saves in computational time. Also, it would be interesting to test additional and lower values to get a more optimized radiation configuration. It can also be optimized by providing part specific values based on whether the part is critical from a radiation perspective.

It is also of interest to perform further comparisons between these setups and the rig. More braking cycles could be simulated, especially cycles in which the disc does not stop. Additional validation using surface-temperature measurements from a thermal camera would also be valuable.

Bibliography

- [1] European Commission. Vehicle emissions and battery durability (euro 7): technical requirements and certification rules. <https://eur-lex.europa.eu/EN/legal-content/summary/vehicle-emissions-and-battery-durability-euro-7-technical-requirements-and-certification-rules.html>, 2023. Accessed: 2026-01-22.
- [2] Volvo Trucks. Who we are. <https://www.volvotrucks.com/en-en/about-us/who-we-are.html>, 2026. Accessed: 2026-01-22.
- [3] California Air Resources Board. Inhalable particulate matter and health (pm2.5 and pm10). <https://ww2.arb.ca.gov/resources/inhalable-particulate-matter-and-health>, 2024. Accessed: 2026-01-23.
- [4] European Commission Joint Research Centre. Particle emissions from tyre and brake wear: On-going literature review summary and open questions. Informal Document GRPE-68-20, Sustainable Transport Unit, Institute for Energy and Transport, Geneva, January 2014. Presented at the 68th GRPE, 7-10 January 2014.
- [5] Y. Wang, A. Li, H. Yin, S. Su, Y. Lai, W. Chen, X. Wang, J. Tan, L. Hao, S. Shuai, and Y. Ge. Brake wear particles from various temperatures: emission characteristics, generation processes and evolutions. *Emerging Contaminants*, 11:100554, 2025.
- [6] J. Morgan. Electric truck regenerative braking, explained. *Fleet Equipment Magazine*, May 2021. Technical feature on heavy-duty electrification.
- [7] V. Chiarelli. What is regenerative braking and how does it work? <https://www.evanshalshaw.com/blog/what-is-regenerative-braking/>, August 2025. Accessed: 2026-02-06.
- [8] J. Sjöblom. Reduction of brake wear emissions from urban heavy-duty transport (reduktion av bromspartikelemissioner från urbana lastbilstransporter). Project Description/Application P2024-03514, Chalmers University of Technology, Gothenburg, Sweden, March 2025. Application within FFI Nollutsläpp.
- [9] R. Limpert. *Brake Design and Safety*. SAE International, Warrendale, PA, 3rd edition, 2011. SAE Product Code: R-398.
- [10] Vaden Team. Detailed analysis of air disc brake caliper systems. <https://www.vadenoriginal.com/en/blog/detailed-analysis-of-air-disc-brake-caliper-systems>, May 2025. Accessed: 2026-01-23.
- [11] A. Mohammad, R. Zamora, and T. Lie. Electric vehicles and smart grids. In *Sustainable Energy Storage in the Next Generation of Smart Grids*, chapter 10, pages 217–244. Elsevier, 2022.

- [12] D. Gross, J. Hauger, W. Schröder, W. Wall, and S. Govindjee. *Engineering Mechanics 3: Dynamics*. Springer Berlin Heidelberg, Berlin, Germany, 2nd edition, 2014.
- [13] B. Andersson, R. Andersson, L. Håkansson, M. Mortensen, R. Sudiyo, and B. van Wachem. *Computational Fluid Dynamics for Engineers*. Cambridge University Press, 2012.
- [14] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide*, 2025. Turbulence.
- [15] R. Rizqyandra. Choosing between k-epsilon and k-omega for your cfd simulation: A practical guide. <https://medium.com/@riqy.sci/choosing-between-k-epsilon-and-k-omega-for-your-cfd-simulation-a-practical-guide-1546d95596ff>, 2023. Accessed: 2026-04-21.
- [16] Siemens Digital Industries Software. Should i use moving reference frames or rigid body motion for my simulation? https://support.sw.siemens.com/en-US/knowledge-base/KB000036324_EN_US, September 2019. Accessed: 2026-03-25.
- [17] Siemens Digital Industries Software. How to accelerate a simulation combining radiation and rigid body motion? https://support.sw.siemens.com/en-US/knowledge-base/KB000038993_EN_US, August 2019. Accessed: 2026-03-26.
- [18] Siemens Digital Industries Software. In a steady state cht simulation, how to deal with the solid motion without moving the solid? https://support.sw.siemens.com/en-US/knowledge-base/KB000032812_EN_US, November 2023. Accessed: 2026-03-26.
- [19] Siemens Digital Industries Software. *Simcenter STAR-CCM+®Documentation*. Siemens Digital Industries Software, 2019. User guide PDF. Cited page 7168. Document ID: 226870983; PL20190920065926087.
- [20] C. P. Kothandaraman. *Fundamentals of Heat and Mass Transfer*. Pearson Education, Inc., 2006.
- [21] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide*, 2025.
- [22] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide*, 2025. Exchanging Heat Transfer Coefficients.
- [23] W. Kaiser and W. Schlachter. *Energy in Plastics Technology: Theory and Practice*. Hanser Publishers, Munich, 2023. p. 58.
- [24] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide*, 2025. Meshing.
- [25] M. Gustafsson, S. Sebben, and A. Vdovin. A coupled approach for vehicle brake cooling performance simulations. *International Journal of Thermal Sciences*, 132:257–266, 2018.
- [26] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide*, 2025. Mapped Contact Interface.
- [27] B. Haddadi, M. Harasek, and C. Jordan. Cost efficient CFD simulations: Proper selection of domain partitioning strategies. *Computer Physics Communications*, 219:121–134, 2017.

- [28] Amazon Web Services. Computational Fluid Dynamics on AWS: AWS Whitepaper. Whitepaper, Amazon Web Services, Inc., March 2022.
- [29] The Engineering ToolBox. Drag coefficient. https://www.engineeringtoolbox.com/drag-coefficient-d_627.html, 2008. Accessed: 2026-02-06.
- [30] The Engineering ToolBox. Rolling resistance. https://www.engineeringtoolbox.com/rolling-friction-resistance-d_1303.html, 2008. Accessed: 2026-02-06.
- [31] Dekati Ltd. Elpi+ – electrical low pressure impactor. <https://dekati.com/products/elpi/>. Accessed: 2026-04-15.
- [32] OICA Working Group on Ducting & Enclosure. Proposal for brake enclosure and sampling for pmp. Internal report / presentation, June 2023. Presentation.
- [33] PTC. Modeling radiative heat transfer. https://support.ptc.com/help/creo/creo_pma/r12/usascii/index.html#page/simulate/cfd/Modeling_RadiativeHeatTransfer.html. Accessed: 2026-05-22.
- [34] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide*, 2025. Data Mappers.

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden
www.chalmers.se



CHALMERS
UNIVERSITY OF TECHNOLOGY