Development of Brake Cooling

Arne Lindgren

Bachelor Thesis in Mechanical Engineering, 15 credits

Halmstad 2016-05-20
Preface

This report is the result of my thesis project, which marks the end of my studies for a Bachelor degree in Mechanical Engineering.

I would like to thank:

- Jon Gunner and his colleagues at Koenigsegg Automotive AB for their cooperation and for providing me with this interesting project.
- The Institution of Mechanical Engineering at Halmstad University for having provided me with office space and access to their simulation computer.
- Senior Lecturer Hans Löfgren, my supervisor, for his support and advice.
- Lars, my brother and tireless mentor, for his advice throughout my university studies.
- My family for their support and especially my girlfriend Pia for bearing with me during this time.

Förord

Denna rapport är resultatet av mitt examensarbete, vilket markerar slutet på mina studier till en Kandidatexamen i Maskinteknik.

Jag skulle vilja tacka:

- Jon Gunner och hans kollegor på Koenigsegg Automotive AB för deras samarbete och för att ha försett mig med detta intressanta projekt.
- Institutionen för maskinteknik vid Högskolan i Halmstad för att ha försett mig med kontorsplats och tillgång till deras simuleringsdator.
- Universitetslektor Hans Löfgren, min handledare, för hans stöd och råd.
- Lars, min bror och outtröttliga mentor, för hans råd under mina högskolestudier.
- Min familj för deras stöd och speciellt min flickvän Pia för att hon stått ut med mig under denna tid.
Abstract

Sports cars need efficient brake cooling as they shall perform well during hard driving conditions, like for example race track driving. Most sports cars use ducts that capture ambient airflow and directs this flow over the brakes to improve the cooling. This project was conducted in cooperation with Koenigsegg Automotive AB and aims to develop more efficient brake cooling ducts for their cars. Computational Fluid Dynamics was used to analyse the convective cooling of the brake disc and the pads. First was the cooling with the previously used ducts analysed in order to establish a reference. Then new concepts were created, analysed and developed in an iterative process. A design is proposed, which have the inlet in the centre of the wheel axle and that directs the air through radial channels to the brake disc. The simulations indicate that the proposed design results in 14% higher heat transfer rate compared to the previously used cooling solution. In addition to the cooling ducts, some passive cooling devices were also simulated. Simulations with these in combination with the proposed design, indicate up to 25% increase in heat transfer rate, but this cannot be fully confirmed due to limitations in the simulation model.
Sammanfattning

Sportbilar behöver effektiv bromskylning eftersom de ska prestera väl under hårda körförhållanden, som till exempel bankörning. De flesta sportbilar använder kanaler som fångar omgivande luftflöde och riktar detta flöde över bromsarna för att förbättra kylningen.

Detta projekt genomfördes i samarbete med Koenigsegg Automotive AB och syftar till att utveckla effektivare bromskylkanaler till deras bilar.

Computational Fluid Dynamics användes för att analysera den konvektiva kylningen av bromsskivan och bromsbelägen.

Först analyserades kylningen med den tidigare använda bromskylkanalen i syfte att skapa en referens. Sedan skapades nya koncept som analyserades och utvecklades i en iterativ process.

En konstruktion föreslås, som har inloppet i centrum av hjulaxeln och som sedan styr luften genom radiella kanaler till bromsskivan. Simuleringarna indikerar att den föreslagna konstruktionen resulterar i 14% högre värmeöverföringshastighet än den tidigare använda bromskylningslösningen.

Förutom kylkanalerna har några passiva kylanordningar också simulerats. Simuleringar med dessa i kombination med den föreslagna konstruktionen, indikerar upp till 25% ökning av värmeöverföringshastigheten, men detta kan inte helt bekräftas på grund av begränsningar i den använda simuleringsmodellen.
# Table of contents

Preface ........................................................................................................................................ II
Förord ......................................................................................................................................... I
Abstract .................................................................................................................................... II
Abbreviations .......................................................................................................................... 1
Nomenclature ............................................................................................................................ 1

1 Introduction ........................................................................................................................... 2
   1.1 Background .................................................................................................................... 2
      1.1.1 Company presentation ............................................................................................. 2
   1.2 Objective ....................................................................................................................... 3
      1.2.1 Problem definition .................................................................................................. 3
   1.3 Delimitations ................................................................................................................ 3

2 Methods ............................................................................................................................... 4
   2.1 Method discussion ......................................................................................................... 4
   2.2 Methods in this thesis .................................................................................................. 4
      2.2.1 Method to analyse air flow and cooling ................................................................. 4
      2.2.2 Concept generation method .................................................................................. 5
      2.2.3 Concept evaluation method .................................................................................. 6
   2.3 Preparations and data collection .................................................................................. 6

3 Theoretical framework ........................................................................................................... 7
   3.1 Brake system description .............................................................................................. 7
   3.2 Brake energy .................................................................................................................. 8
   3.3 Temperature calculation ............................................................................................... 10
   3.4 Aerodynamics .............................................................................................................. 10
      3.4.1 Boundary layer ...................................................................................................... 10
      3.4.2 Flow type ............................................................................................................. 11
      3.4.3 Computational Fluid Dynamics .............................................................................. 12
   3.5 Modes of heat transfer ................................................................................................. 12
      3.5.1 Conduction ........................................................................................................... 13
      3.5.2 Convection ............................................................................................................ 13
      3.5.3 Radiation .............................................................................................................. 14
   3.6 Summary of relevant literature .................................................................................... 15

4 Component description ....................................................................................................... 16
   4.1 Components ................................................................................................................ 16
      4.1.1 Brake disc ............................................................................................................. 16
      4.1.2 Brake pads ............................................................................................................ 17
      4.1.3 Brake caliper ....................................................................................................... 17
Abbreviations

CAD          Computer Aided Design
CFD          Computational Fluid Dynamics
C/SiC        Carbon fibre-reinforced silicon carbide
CPU          Central Processing Unit
FEM          Finite Element Method
MRF          Multiple Reference Frame

Nomenclature

$A_f$        Frontal area
$a$          Acceleration
$c$          Specific heat capacity
$C_D$        Coefficient of drag
CG           Centre of gravity
$F$          Force
$h_c$        Convective heat transfer coefficient
$k$          Thermal conductivity
$L$          Length
$m$          Mass
$M$          Torque
$N$          Normal force
$Q$          Heat energy
$q$          Heat energy transferred per unit time
$T$          Temperature
$t$          Time
$v$          Velocity
$W$          Energy (work)
$\mu$        Friction coefficient
$\rho$       Density
1 Introduction

1.1 Background
Automotive brakes generate a considerable amount of heat during braking. This heat energy needs to be sufficiently dissipated before the next braking event, otherwise the temperature in the brake components will increase. This may still be within the thermal limit of the brakes, but if the inflow of heat energy over time is greater than the dissipation rate then the brakes will overheat. This is usually not an issue for ordinary road cars, because the cooling from the ambient air flow is normally sufficient with regard to the heat generation rate during driving. Sports cars have higher performance and are more likely to be used for race track driving and similar conditions that places a higher demand on the brakes. Therefore high performance sports cars and racing cars often have cooling ducts that channels air to the brakes in order to improve the cooling.

1.1.1 Company presentation
Koenigsegg Automotive AB is a Swedish company that was started in 1994 and is a relatively small but prominent manufacturer of extreme and technologically advanced sports cars. The company is since 2003 based at the former air force wing F10 outside Ängelholm, Sweden.
Their most recent model, the Koenigsegg Regera, was launched at the Geneva motor show in March 2015. The Regera is planned to go into production during the second quarter of 2016 and it is the first Koenigsegg to utilize hybrid technology. The combined power of its internal combustion engine and its electric motors exceeds 1500 horsepower.

![Koenigsegg Regera](Koenigsegg Automotive AB, 2015)
1.2 Objective

The objective of this project is to develop more efficient brake cooling ducts for the front brakes of the Koenigsegg Regera.

1.2.1 Problem definition

The high performance of the Koenigsegg Regera requires efficient brakes. The average brake power during braking from 300 to 0 km/h, is about 1 MW. Experience from previous models show that track driving can result in excessively high brake temperatures. This may lead to friction material degradation, damage to the brake discs or brake fluid vaporisation. These failure modes can potentially result in partial or complete loss of braking. It can also lead to thermal damage to surrounding parts.

To aid the cooling of the brakes, airflow from the front of the car is captured with ducts and channelled to the brakes through a flexible hose. This brake cooling hose is attached to an air guide (or brake duct) that is mounted to the wheel bearing carrier, and directs the air flow against the brakes. In order to refine the design of the brake ducts, the air flow and heat transfer process in this particular case needs to be investigated.

1.3 Delimitations

This project focuses only on improving the convective air cooling of the front brakes. The proposed brake cooling solution is only developed to a feasible concept, no final design is made.
2 Methods

2.1 Method discussion

When investigating a quantitative engineering problem, there are three main methods: analytical methods, numerical methods and experiments. Many engineering problems can be solved with analytical calculations, and it is a method normally used at some stage in most projects. Even if it is frequently necessary to make assumptions and simplifications to the problem to be able to calculate it, it can normally provide valuable answers. However the analytical approach is only suitable for simple models. For more complex models like fluid flow involving non-trivial three-dimensional geometry, no general analytical solution has been found. (See chapter 3.4)

Numerical simulations are run on computers which perform a huge number of calculations and give an approximate solution. The approximation can however be very good if the calculation is detailed enough. The computational possibilities have increased dramatically in the last decades, mainly due to advances in computing power and software development. For example, fluid flow simulations were performed almost exclusively in the aerospace industry until some decades ago. Currently several commercial and some freeware fluid flow simulation software's are available. The numerical approach makes it possible to study the subject in detail and monitor the properties of interest. On the other hand, it can be difficult to replicate the reality in a simulation and if the physical process is not fully understood (as with fluid flow), there is some lack of confidence in the results.

The experimental approach is often the most accurate since few assumptions and simplifications are needed. However, in some cases it may be difficult to accurately measure physical properties with good precision. In such cases it is hard to interpret the results and draw conclusions for further development.

An experimental approach is usually more expensive and requires more resources, than analytical or numerical methods do.

2.2 Methods in this thesis

2.2.1 Method to analyse air flow and cooling

This project involves air flow around non-trivial geometry in combination with heat transfer, therefore the analytical approach is not applicable. An experimental approach would be possible but it would require a great number of prototypes, expensive test equipment and the execution of many tests. It would also be difficult to measure properties like air velocity and temperature in detail, it would only yield results on a number of points. Therefore the understanding of the cooling process would be weak.

The numerical approach was chosen for this project because it is the only feasible method available. The branch within numerical methods for flow simulations is
called *Computational Fluid Dynamics (CFD)* and that was the tool used in this project.

### 2.2.2 Concept generation method

The generation of concepts in this project is mainly according to the method described by Ullman (2010), see Figure 2.1. A first batch of concepts are generated using the limited knowledge available, these are then evaluated and analysed. The knowledge gained from the analysis is used to make changes and refinements. New ideas are also introduced in response to problems seen in the earlier concepts. This method can also be called an iterative design process. The majority of this project has been spent in the upper portion of the flowchart in Figure 2.1.

![Figure 2.1: The conceptual design phase (Ullman, 2010)](image-url)
2.2.3 Concept evaluation method

First was the previous brake cooling solution (designed by Koenigsegg) simulated in order to obtain a reference. This reference is referred to as the baseline in this report. Then the simulation results of the developed concepts were compared to the baseline using quantitative cooling performance data. In addition to this, generated contour and vector plots were used to analyse flow properties and temperature in order to understand the effects of the different concepts. Such understanding is necessary to be able to generate new improved designs. Design criterias were also defined and were used to select the most appropriate concept.

2.3 Preparations and data collection

A review of relevant research about brake cooling was carried out, as well as studies in CFD-modelling. Interviews were used as the primary method to collect information about the design criteria's, the background and available measurement data. CAD models of the relevant geometry were received as well as air flow plots of the complete car model from CFD simulations that Koenigsegg had conducted.
3 Theoretical framework

3.1 Brake system description

Disc brakes are almost the only mechanical brake type used in contemporary cars. The brake consists of a disc (hence the name) which is coupled to the wheel and have a stationary brake caliper which presses the brake pads against either side of the disc during braking. The brakes on passenger cars are actuated with hydraulics using a special hydraulic fluid (brake fluid) in order to avoid vaporization at high temperatures.

Regenerative braking is a technology that has become more and more common in recent years. Regenerative braking is when the brake energy is recovered and reused, for example with a Motor Generator Unit (MGU). This technology is however not sufficient as the only brake system, mechanical brakes are still necessary as backup and for heavy braking such as in an emergency situation.

![Hydraulic disc brake system](www.buildyourownracecar.com)

*Figure 3.1: Hydraulic disc brake system (www.buildyourownracecar.com)*
3.2 Brake energy

Mechanical brakes are transforming kinetic energy to heat energy and usually an insignificant amount of energy is also dissipated as sound. During a brake event, the brake caliper pistons presses the brake pads against the surfaces of the rotating brake disc, resulting in friction between the pads and the disc. The friction force results in a counteracting (braking) torque due to the distance from the rotational centre. Heat is generated due to the friction in the sliding contact between the pads and the disc. The amount of energy converted to heat in the brakes during a braking event can be calculated using the formulas below.

The kinetic energy absorbed during the braking event can be calculated using formula 3.1:

\[ W_K = \frac{1}{2} m (v_0^2 - v_1^2) \]  

(3.1)

Apart from the braking deceleration, the car is also decelerated by the aerodynamic drag force, formula 3.2:

\[ F_D = \frac{1}{2} \rho v^2 C_D A_f \]  

(3.2)

There are also other losses, like rolling resistance, but these are negligible in this case. In order to get the work done by the drag force, it is multiplied by the distance in the form of \( v \, dt \). And since velocity is not constant during the braking event, it must be defined as a function of time \( v(t) \). The work due to drag are obtained by integrating over the duration of the braking event.

\[ W_D(t_0) = \int_0^t F_D \, v(t) \, dt \]  

(3.3)

The total energy (or work) that the mechanical brakes convert to mainly heat, can be calculated by subtracting the drag energy from the kinetic energy:

\[ W_{MB} = W_K - W_D \]  

(3.4)
The amount of energy absorbed by the front brakes is proportional to the load on the front axle if the tyre friction coefficient is the same front and rear. With the assumption that the car is braked at the limit of lock up, that there is negligible aerodynamic downforce and that the aerodynamic drag force acts through the CG, the load proportion on the front axle can be calculated as below:

\[ F_f = \mu N \]  
\[ F_f = \mu N \]  
\[ \sum F_x = 0 = N_1 + N_2 - mg \]  
\[ \Rightarrow N_1 + N_2 = mg \]  

\[ \sum m \alpha = 0 = ma - F_f - F_f = ma - \mu (N_1 + N_2) \]  
\[ \Rightarrow a = \mu g \]  

\[ \sum M(2) = 0 = N_1 (L_1 + L_2) - mg L_2 - ma L_3 \]  
\[ \Rightarrow N_1 = \frac{mg (L_1 + \mu L_3)}{(L_1 + L_2)} \]  

Load proportion front axle:  
\[ \frac{N_1}{mg} = \frac{L_2 + \mu L_3}{L_1 + L_2} \]  

So the amount of energy that is converted to heat in one front brake can be calculated with formula 3.11:

\[ W_{mbf} = \frac{1}{2} W_{mb} \frac{L_2 + \mu L_3}{L_1 + L_2} \]  

\[ \text{Figure 3.2: Free body diagram of a braking car} \]
3.3 Temperature calculation

If heat energy is added to a substance, its temperature (internal energy) increases. The magnitude of the temperature raise depends on the material property specific heat capacity \( c \) and the mass of the object. The temperature change is calculated with the formula:

\[
\Delta T = \frac{Q}{cm} \tag{3.12}
\]

where

\( \Delta T \) = Temperature change [K]
\( Q \) = Heat energy transferred [J]
\( c \) = Specific heat capacity \([\frac{J}{kg \cdot K}]\)
\( m \) = mass [kg]

3.4 Aerodynamics

The brakes are cooled with airflow so in order to improve the cooling, some knowledge of aerodynamics is necessary. Aerodynamics is the sub-field of fluid mechanics that is focused on the science of how air flows and interacts with solid objects. Traditionally aerodynamics has been studied using experimental methods, mostly by the means of wind tunnels. This is because the governing equations that are valid for any kind of flow, the Navier-Stokes equations, are very difficult to solve for other than very simple theoretical cases. No general analytic solution has been found for this set of non-linear partial differential equations despite the fact that they were formulated at end of the 19th century (Clay Mathematics Institute, 2000).

Today, numerical methods are used to calculate simplified Navier-Stokes equations, which have become feasible because of the development of powerful computers. Wind tunnel studies are however still necessary in many cases for validation purposes.

3.4.1 Boundary layer

When fluid flows over a surface, the fluid molecules close to the surface have no velocity (in the surface reference frame) due to the friction between the surface and the fluid molecules. The velocity of the fluid increases with the distance from the wall until it is the same as the freestream velocity. This layer of fluid with a velocity gradient is called the velocity boundary layer.

If there also is a temperature difference between the surface and the fluid, there will be a temperature boundary layer. Similarly, in the temperature boundary layer, the
fluid temperature changes with the distance from the surface. The thickness of these two boundary layers is usually defined as the distance from the surface to where the property (velocity or temperature) is 99% of the freestream property.

3.4.2 Flow type

At low velocities the flow close to a surface is laminar, which means that the fluid flows in "streamlines" without any mixing or swirls. At higher velocities the flow near the surface becomes turbulent, which means that the fluid is mixing and has chaotic swirls and eddies. Turbulence occurs when the inertia forces dominate over the viscous forces in the fluid. The Reynolds number (Re) is a dimensionless number which is used to determine the flow type. A low Reynolds number means laminar flow and high numbers generally means turbulent flow.
3.4.3 Computational Fluid Dynamics

With CFD, the fluid motion is solved approximately in local cells. The Navier-Stokes equations are simplified with approximations in order to be able to solve them with numerical methods. Different approximation technics are used, such as Reynolds Averaged Navier-Stokes (RANS) or Large Eddy Simulation (LES). Numerical methods use interpolation and gives approximate answers at discrete points. Consequently, the fluid volume that should be simulated (called computational domain) must be divided into a finite number of cells/elements. The process of defining the computational cells is called meshing. A denser mesh results in higher precision, because of the interpolation that is done. The Finite Volume Method (FVM) is the most common numerical method employed by CFD-software, the Finite Element Method (FEM) is also used to some extent. In cases where turbulence is present (most real cases), a turbulence model is required. The ability to calculate heat transfer and other processes is also available in some CFD software's. Mainly due to the difficulties to model turbulence (a phenomenon that is not fully understood), the results of CFD simulations should be used with some caution. CFD is however a rapidly evolving technology and the subject of much research.

3.5 Modes of heat transfer

Heat is generally dissipated in three ways, Conduction, Convection and Radiation. This is illustrated in Figure 3.5, which uses a disc brake as an example.

![Figure 3.5: Brake heat transfer modes (Limpert, 1999)](image-url)
3.5.1 Conduction

Conduction is the heat transfer process that takes place in solids and in stationary fluids, or through physical contact between substances. This heat transfer process takes place at a molecular level, the heat energy causes the atoms to vibrate and collide with neighbouring atoms resulting in a domino effect that propagates throughout the substance. In for example metals also free electrons dissipate the heat. Solids and in particular metals are good heat conductors due to their closely packed atoms while gases are less conductive due to larger distances between molecules. The empirical formula for conduction also known as Fourier's Law:

\[ q = -k A \frac{\Delta T}{dx} \]  \hspace{1cm} (3.13)

where

- \( q \) = heat energy transferred per unit time \([W]\)
- \( k \) = thermal conductivity of the material \([\frac{W}{mK}]\)
- \( A \) = surface area \([m^2]\)
- \( \Delta T \) = temperature difference over the material thickness \([K]\)
- \( dx \) = thickness of the material \([m]\)

3.5.2 Convection

Convection is a heat transfer process that only takes place in fluids and involves the movement of fluid molecules that transports the thermal energy. If the fluid movement occurs due to density variations caused by local temperature gradients in the fluid, it is called natural or free convection. This is the way a heating element normally works in a room, the air close to the element is heated (through conduction) and then that hot air rises due to it being lighter than the surrounding air. This air motion helps to dissipate the heat in the room and continuously pass cold air over the heating element. If the fluid motion is caused by an external force such as wind or a fan it is called forced or assisted convection. Forced convection is in many applications used to increase the rate of heat exchange.

The empirical formula for convection, also known as Newton's Law of Cooling:

\[ q = h_c A \Delta T \]  \hspace{1cm} (3.14)
where

\( q \) = heat energy transferred per unit time [\( \text{W} \)]

\( h_c \) = convective heat transfer coefficient [\( \text{W}/(\text{m}^2\cdot\text{K}) \)]

\( A \) = surface area [\( \text{m}^2 \)]

\( \Delta T \) = temperature difference between the surface and the bulk fluid [\( \text{K} \)]

The convective heat transfer coefficient \( (h_c) \) depends on several fluid factors such as type of fluid, velocity, turbulence and viscosity. Therefore, the convective heat transfer coefficient is not a constant and it is difficult to determine for other than simple cases in controlled experimental environments.

But in general, these factors should be optimized to improve convective heat transfer:

- Maximize velocity (velocity between surface and fluid)
- Maximize fluid turbulence in the boundary layer
- Maximize \( \Delta T \) (difference between surface and bulk fluid temperature)
- Maximize surface area
- Maximize fluid conductivity (i.e. choice of fluid)
- Minimize fluid viscosity

3.5.3 Radiation

Radiation is a heat transfer mode that consists of electromagnetic waves (similar to light) that are emitted by the heated object. Unlike conduction and convection which needs a medium to transport the heat energy, radiation also occurs in vacuum and can travel vast distances. Air and many other gases are practically transparent for radiation, which means that little or no energy is absorbed by it. The electromagnetic waves emitted during normal cooling are in the so called infrared spectrum, which is invisible for the human eye. At elevated temperatures the radiation can enter the visible spectrum, like melted steel for example.
3.6 Summary of relevant literature

A lot of research has been done on the subject of automotive brake cooling. However, most studies are focused on the design of brake disc ventilation channels rather than the utilization of ducted airflow to cool the brakes. Many of these studies of brake disc channels are also performed in isolation without the influence of the wheel rim and the external airflow. Stephens (2006) investigated experimentally the influence of local aerodynamics for the dissipation of brake heat, using a brake test machine in a wind tunnel. His investigation confirmed that the vehicle speed has the strongest influence on the brake heat dissipation. It was however found that the airflow through the rim at high vehicle speeds can severely disrupt the airflow through the disc ventilation channels. The second most influencing factor was found to be the rim design, how much the spoke design restricted the airflow. Neys (2012) studied the brake heat generation and dissipation in order to develop a brake temperature calculation model for an in-car warning system. The calculations performed in that case showed that 98.8% of the friction heat energy is absorbed by the brake disc. In the paper of Talati & Jalalifar (2009) it is stated that 93.4% of the generated heat is absorbed by the brake disc. This difference is likely to be due to different material properties or calculation approximations. Thuresson (2014) performed correlation studies of convective brake heat cooling between CFD and wind tunnel tests as well as CFD brake disc simulations with full vehicle model versus isolated brake disc. Different modifications of a standard brake shield were also studied in order to determine how much they affect the cooling. It was concluded that there were considerably different flow in the disc cooling channels with the wheel and caliper in place. In this study the brake disc channels accounted for the biggest convective heat transfer from the disc. The brake shield modifications showed that the shield has a considerable impact on the brake disc cooling, especially because it affects the supply of air to the disc ventilation channels. In the research paper by Kiran (2015) different ways of directing the external airflow through the wheel in order to improve cooling is investigated with CFD. However only two concepts were analysed and the concepts were mainly aimed at increasing the overall through-wheel flow and no study was performed on how the air should be directed within the wheel to optimize the cooling. The automotive industry and in particular the racing industry has done a great deal of research in this area but most information is kept undisclosed in order to gain an advantage over their competitors.
4 Component description

The components of the front brakes and the relevant parts of the suspension are presented in this chapter. The geometry of these components entails constraints for the brake duct concepts since their design is already finished and outside the scope of the project. The previous brake cooling solution (the baseline) which this project aims to improve is also presented.

4.1 Components

4.1.1 Brake disc

The front brake discs are made of Carbon fibre-reinforced silicon carbide (C/SiC) and are 396 mm in diameter and has a thickness of 38 mm. Ventilation channels are milled in a branched formation with ten inlets at the inside diameter and with 30 outlets at the outer diameter. The brake disc also has 60 cross drilled (axial) 6mm holes that pass through the ventilation channels. The disc is mounted to its stainless centrepiece with a floating design, which allows the disc to move a small distance in the axial direction and expand radially while it is kept centred about the rotational axis. Between the bolts holding the stainless centrepiece to the disc there are axial gaps approximately 2 mm wide.
4.1.2 Brake pads

The brake pads are of a conventional design, with a steel back plate to which the friction material is bonded. The friction material has low thermal conductivity compared to the brake disc material.

Figure 4.2: Front brake pad

4.1.3 Brake caliper

The brake calipers are of the fixed type with a total of six pistons, three on each side of the disc. The caliper body is milled from aluminium in two pieces which are bolted together. The pistons are made of a ceramic material in order to conduct less heat into the brake fluid.

Figure 4.3: Front brake caliper

4.1.4 Upright

The upright or wheel bearing carrier is the load bearing structure between the wheel bearing and the wishbones and steering rod. It is a milled aluminium part.

Figure 4.4: Upright
4.1.5 Wheel axle

The wheel axle is a machined steel part and it is hollow to reduce the weight.

![Figure 4.5: Wheel axle](image)

4.1.6 Wheel rim

The Koenigsegg Regera wheel rims are made of carbon fibre to reduce the unsprung weight. They are 19 inch in diameter.

![Figure 4.6: Regera wheel rim](image)
4.2 Baseline brake duct
This brake cooling duct has been used in previous Koenigsegg models and it is used as the baseline in this project. It is made of carbon fibre composite and it is designed like a nozzle that directs the hose airflow towards the centre of the brake disc.

Figure 4.7: Brake cooling duct mounted on Agera upright. (Klingelhofer, 2013)
5 Project execution

In order to be able to compare the conceptual brake duct designs with the baseline, the first step of the project was to simulate the baseline. Values of heat transfer rates at the different brake surfaces were obtained for use as reference later on. Next step was to generate brake duct concepts and simulate their effect on the cooling. The same computational model was used for both baseline and concept simulations, only the brake duct geometry and naturally the position of the duct inlet boundary condition were changed.

5.1 Simulation software

Several CFD codes are available for use on desktop computers. There are even some that are released as free open source software. Many of these are targeted at specially trained CFD engineers and are difficult to operate for the novice user. The software *FloEFD for CATIA V5* from MentorGraphics was chosen for this project. This choice was made mainly due to the availability of the software at the institution but also because it is easily operated by the novice CFD user and that it is embedded within the CAD system *CATIA V5*. The simulation software's from MentorGraphics have historically been aimed at electronics design, but the FloEFD software was added to their portfolio in 2008 after the acquisition of the Flomerics company (Schnitger Corporation, 2010). With traditional CFD-software, the 3D-model needs to be exported from the external CAD-system and imported in the CFD software. This makes the work inefficient due to the importing/exporting that has to be executed every time the geometry needs to be altered. With the chosen software the configuration of the simulation is made from within the CAD software, which eliminates the need for exporting/importing geometry. MentorGraphics calls this technology *concurrent CFD*. The software also has features such as automatic meshing, case configuration wizard and post processing, all from within the CAD software. FloEFD utilizes a Cartesian mesh, which means that the mesh is composed of rectangular cuboid elements that are aligned with the global coordinate system. When applying mesh refinements these global cells are divided \(n\) times along each direction until the desired local refinement level is reached. This means that the dimension of the refined cell is \(1/2^n\) of the global cell size. When a cell is intersected by the boundary of a solid, it is split by a plane which defines the fluid control volume versus the solid control volume. The configuration of the computational model also includes setting up boundary conditions and calculation goals. A goal is a quantity of interest and it can be specified as a global, volume, surface or point property. The software uses the goals to refine the calculation for the relevant quantities, for determining when the calculation should finish and also saves the calculation history of these values.
5.2 Computational Model

The choice of model complexity and simplifications are important in order to obtain a working and manageable simulation while still capturing the relevant physics of the task.

5.2.1 Geometric model

Geometrical models with different scope were simulated and evaluated. The normal cooling, due to ambient airspeed and venting from wheel rotation, was important to include in the simulation. Otherwise the result could be a solution which in reality results in inferior cooling because of the shielding effect of the duct. It was concluded that a minimal model with only the geometry of the brake disc and duct was insufficient, as the normal cooling cannot be realistically replicated. The wheel and the wheelhouse are particularly important as they have strong influence on the flow behaviour around the brakes. A reasonable calculation time was however needed, in order to be able to simulate numerous different concepts. Generally, the calculation time increases with increased size of the computational domain, because of the number of cells needed. Therefore it was not possible to use the full vehicle body in the simulations. It was decided to utilize a relatively small computational domain with a geometrical model consisting of a partial car body with wheelhouse, the wheel and brake assembly (see Figure 5.1). The wheelhouse has similar dimensions and ground clearance as on the Regera. The external shape of the partial car body was modelled as a box parallel to the domain boundary and with only the bottom surface and the side surface inside the computational domain. This yields a constant blockage ratio which minimizes the risk of numerical errors due to the small computational domain. The external shape of the body has low significance as it has little influence on the flow condition close to the brake disc.

![Figure 5.1: The geometrical model (blue arrow indicates ambient airflow direction)](image)

21
5.2.2 Vehicle speed

To simulate the cooling dynamically with changing vehicle speed and temperatures etc. would be too complex and computationally expensive. Due to this it was decided to conduct the simulations at a fixed speed. The brake cooling is mainly needed during race track driving, and since the absolute majority of the cooling takes place between the braking events, it is logical to optimise the cooling for the typical average speed around a lap. It was decided that the simulations should be conducted at a vehicle speed of 150 km/h as this is a typical average speed on a race track. The cooling should of course also work at both lower and higher speeds. The brake cooling hose flow rate that was specified by Koenigsegg at a vehicle speed of 250 km/h was scaled according to the assumption that the hose flow rate is proportional to vehicle speed, which should be reasonably accurate.

5.2.3 Rotating parts

The rotational speed of the wheel and brake disc are modelled with two methods, with wall conditions for all rotationally symmetric surfaces (see Figure 5.2) and with local rotating regions (see Figure 5.3) for non-rotationally symmetric geometries such as rim spokes and brake disc channels. The rotating regions are modelled with the sliding mesh approach which requires the simulation to be time-dependent (i.e. transient). A less computationally expensive way would be to use the averaging approach for rotating regions, which would allow the simulation to be run as steady state. With this approach, all flow past the rotating region boundary is circumferentially averaged. But because of the averaging, it doesn’t simulate the non-axisymmetric flow that is present in this case correctly. This is because all flow through the region boundary will show up as the net flow on the other side of the boundary (i.e. all circumferentially local in/out flows and turbulence will not be present after the passage of the boundary.) Some CFD codes feature a method called Multiple Reference Frame (MRF) to simulate rotating parts in steady-state, this method is however not available in FloEFD.
Figure 5.2: Wheel and ground boundary conditions

Figure 5.3: Rotating regions (highlighted in turquoise)
5.2.4 Simplifications

Convection is the dominating and most effective heat dissipation process in a typical automotive disc brake system, research has shown that 60-90% of the heat is dissipated through convection while the rest is dissipated through radiation and conduction (Stephens, 2006). In this particular case with a floating brake disc, which results in a very small contact area between disc and disc-centre, the conduction portion will be even lower. Radiation is quite difficult to simulate because all the model surfaces needs to be assigned absorption, reflectivity and transmissivity properties and these are difficult to determine. Convection is the heat dissipation process that is easiest to affect. Due to these facts and to obtain a simpler model, only convection is considered in the simulations.

The cross drilled (axial) holes in the brake disc had to be removed in order to enable the use of a local rotating region to simulate the rotation of the disc channels. This is because the interface between the rotating region and the surrounding geometry needs to be rotationally symmetric. It was also not possible to put the whole brake disc inside the rotating region, because the close proximity of the brake pads would make the region interface non axisymmetric. The removal of the holes will only have a small effect on the accuracy of the simulation since the holes are quite small (6 mm) and that the flow through them doesn’t have a big influence on the overall cooling of the disc.

5.2.5 Boundary conditions and Wall conditions

Since conduction is not considered in the parts, the heat is modelled as fixed temperatures on the surfaces of the parts where the heat is generated (disc and pads), see Figure 5.4. The surface temperatures are based on the values recorded by Koenigsegg during track testing.

![Figure 5.4: Disc and pad wall conditions](image-url)
All other model surfaces are assigned the default wall condition, adiabatic, which means that they neither absorb nor emit heat. This is of course a simplification but it has low influence on the heat transfer rates at the heated surfaces.
The flow field entering the computational domain is specified to have a uniform speed of 150km/h. On the inboard side of the wheelhouse there is also an incoming flow from the radiator (see Figure 5.1). This mass flow was estimated (using CFD results from full vehicle simulations) to be 1 kg/s.
The cooling hose flow rate that is valid for the baseline duct was obtained from a full vehicle CFD simulation performed by Koenigsegg. But other duct designs can generate different pressure drop or flow resistance, and consequently result in another flow rate. Therefore it was decided to model this flow as a pressure at the duct inlet. This approach results in a more accurate flow rate for other duct design, then if a fixed flow rate was used. The total pressure (static + dynamic) to apply was determined by simulating the baseline duct with the flow rate that was obtained from Koenigsegg (scaled to 150 km/h).
The temperature of the ambient air and the hose air flow was specified as 20°C while the flow from the radiator was specified as 30°C. See appendix 1 for complete CFD set up specification.

5.2.6 Turbulence settings

The simulation flow type was specified as *laminar and turbulent*, which means that FloEFD solves the flow problem with both flow types and deals with the transition between the two.
The k-ε turbulence model available in FloEFD was used with the default values, turbulent energy 1 J/kg and turbulent dissipation rate 1 W/kg. It is often difficult to estimate the turbulence in advance, therefore the default values are recommended (MentorGraphics, 2014a).
5.2.7 Computational Mesh

The configuration of the computational mesh is sensitive since it needs to be fine enough to enable accurate results as well as it should not yield a too computationally expensive simulation.

In order to verify that the mesh was fine enough, mesh convergence tests were performed. This is done by performing several simulations with identical set up except that the mesh cell size is refined for each simulation run. The physical parameter of interest (in this case the heat transfer rate) is plotted on a chart with one marker for each run. The mesh refinement level is considered adequate at the point where the results converge (i.e. don't change considerably with cell size). This was found to be difficult in this case as the convection heat transfer process needs a very fine mesh in the boundary layer of the heated parts to get an accurate result. The computational resources were a limiting factor that prevented further mesh refinement even though the mesh was only refined in the most desired regions. FloEFD is however utilizing a two scale wall function technique that solves the flow in boundary layers with different methods depending on how many cells there are across the boundary layer thickness. This solves the sub-cell boundary layer flow with a special integral method which enables the use of a coarser mesh than would be necessary otherwise (MentorGraphics, 2011).

A satisfactory mesh refinement level with manageable calculation time could despite this not be found. Therefore a compromise between precision and calculation time had to be made. But as the error (in heat transfer rate) most likely is rather consistent, the results can be used for direct comparisons between different duct designs. Figure 5.5 shows the constructed mesh which consists of approximately 3,5 million cells.
Figure 5.5: Computational mesh
5.2.8 Initialization

In FloEFD it is possible to use the results from a previous simulation as the initial state for another simulation, this reduces the necessary calculation time because the main flow field is already developed. A simulation was run with no hose flow or brake duct but all other settings equal, and was used as initial state for all simulations.

5.2.9 Calculation

A transient simulation uses a time-step to iterate towards a converged solution, the time-step was specified so that each step was equal to one degree of rotation. This is adequate for this type of case according to the FloEFD technical reference manual (Mentor Graphics, 2014). The analysis interval is the number of iterations that the goal result values are averaged over. It was set at 120 iterations (i.e. degrees) in order to average over the variations occurring due to the threefold rotational symmetry of the rim. A transient analysis with rotating parts that are non-rotationally symmetric like in this case never really converges due to the oscillations originating from the rotation. Therefore a specified finishing criterion was prepared so that the finishing moment of the simulation could be automated. The simulation was specified to finish when the change in heat transfer rates over the analysis interval was below 10% of the value at the latest iteration (see Figure 5.6). This enables the simulations to be stopped at a comparable level of convergence automatically. This set-up results in a simulation time of approximately 24 hours on a computer with one six-core Intel Xeon E5 CPU at 3.5 GHz and 32GB RAM.

![Figure 5.6: Goal convergence graph](image)
5.3 Design considerations

In order to make sure that the developed solution satisfies the needs, criterias were formulated according to the results of the interviews. The obvious criterion is that the design should improve the cooling. The design needs to withstand the forces, vibrations, temperatures etc. that occurs in this application. It is also desired that the design protects the carbon fibre wheel rim from excessive heat. A problem with the current design (with the hose attachment in front of the upright) is that there is quite a lot of movement in the cooling hose when the wheels are steered. This movement should be minimized, which is accomplished if the hose is attached close to the wheel steering axis. I should also be as light as possible as everything on a sports car.

Some of these criterias are not particularly relevant since they relate to the final stages of the design process, which are not conducted in this project.

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Improve brake cooling</td>
<td>Requirement</td>
</tr>
<tr>
<td>Protect wheel from high temperature</td>
<td>Goal</td>
</tr>
<tr>
<td>Low weight</td>
<td>Goal</td>
</tr>
<tr>
<td>Durable (heat, vibration, oxidation etc.)</td>
<td>Goal</td>
</tr>
<tr>
<td>Hose attachment with small displacement during cornering</td>
<td>Goal</td>
</tr>
</tbody>
</table>

The friction faces of the pad and disc are of course the hottest areas since the heat is generated there. The brake pads are relatively small and because of their material properties, they don't absorb very much heat energy. Due to this, the temperature gradient in their thickness direction is normally quite big. The majority of the friction heat is absorbed by the brake disc (Neys, 2012; Talati & Jalalifar, 2009). The brake disc friction faces are large and exposed, which means that they are easy to cool. The friction heat is also conducted through the disc thickness to the channels. Usually the disc channels accounts for the largest portion of the heat transfer from the disc (Thuresson, 2014). However, one of the most serious consequences of overheated brakes are brake fluid vaporization, which is due to high caliper/piston temperature. So, one possible strategy would be to focus the cooling on the caliper. It is however better to prevent the root cause than to treat the symptoms. It was therefore logical to mainly focus the cooling attempts at the brake disc.

The main technique used to increase the convective cooling is to break up the temperature boundary layer, either with high air velocity or by introducing turbulence.
5.4 Concept generation

The concepts were developed using own ideas, combined with brake cooling designs observed during earlier work experience and seen in other applications. Because of the relatively long simulation time it was important to try as many features in each concept as possible. Although, it was essential to be able to distinguish the effect of each feature. Some consideration of the manufacturability in appropriate materials were taken during the concept generation phase, so that impossible solutions was avoided. Apart from that, the concepts were not adapted for a specific manufacturing method or designed with necessary brackets etc.
6 Results

First are the results from the baseline simulation presented, then are the results of the generated concepts presented. The primary result is the heat transfer rates, the secondary result, the mass flow rates are only to gain understanding. Contour plots showing air velocity and temperature are also shown. The brake duct designs are highlighted in blue for clarity.

6.1 Baseline simulation result

<table>
<thead>
<tr>
<th>Description</th>
<th>Heat transfer rate [W]</th>
<th>Mass flow rate [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description: The design used on previous Koenigsegg models. Nozzle that blows towards the axle centre. The attachment of the cooling hose is shown in Figure 4.7.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Result:</td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td></td>
<td>14481</td>
<td>2837</td>
</tr>
</tbody>
</table>
RESULTS

Baseline contour plots

Figure 6.1: Velocity, horizontal plane through wheel axis.

Figure 6.2: Velocity, vertical plane through wheel axis.

Figure 6.3: Velocity, plane through brake disc channels.

Figure 6.4: Temperature, horizontal plane through wheel axis.
6.2 Concept results

The majority of the developed concepts and their results are presented in this section, a few more where simulated but are not included in this report. The results are given as the difference in percent from the baseline value.

<table>
<thead>
<tr>
<th>Concept</th>
<th>Description/Idea: Inlet between upright and caliper, blowing over inner brake pad before the air is directed into the centre of the disc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total Disc centre</td>
<td>Disc channels</td>
</tr>
<tr>
<td></td>
<td>-1,4%</td>
<td>-14%</td>
</tr>
</tbody>
</table>

| Comment/Analysis: Uneven cooling on pads, the inlet is smaller and more restricting then the baseline. No overall improvement. |
**RESULTS**

### Concept 2

| Description/Idea: | Inlet in front of upright, all air is directed into the centre of the disc in order to increase flow in disc channels. |

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td>1,1%</td>
<td>-4,8%</td>
<td>2,5%</td>
</tr>
</tbody>
</table>

| Comment/Analysis: | Only small cooling improvement in disc channels, channel flow lower than baseline. Very small overall improvement. |

### Concept 3

| Description/Idea: | Inlet in front of upright, the air is distributed around the disc inner diameter and the air then flows mainly radially along the inside disc surface. The centre of the brake disc is isolated from the hose flow with walls and the disc channels aspirate through the opening between the caliper and upright. The idea is that it shall create a cooling flow over the inner brake pad. |

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td>2,7%</td>
<td>-28%</td>
<td>-24%</td>
</tr>
</tbody>
</table>

| Comment/Analysis: | Very good cooling of the inside disc surface compared to the baseline, but worse almost everywhere else. Low flow rate in disc channels, probably due to inlet restriction. |

(Brake disc removed)
RESULTS

Concept 4

Description/Idea:
Similar to concept 3, but with less restriction to the centre to improve disc channel flow/cooling.

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total</td>
<td>Disc centre</td>
<td>Disc channels</td>
</tr>
<tr>
<td>-0.3%</td>
<td>-39%</td>
<td>-27%</td>
</tr>
</tbody>
</table>

Comment/Analysis:
No improvement compared to Concept 3. Even lower disc channel flow rate and heat transfer.

Concept 5

Description/Idea:
Similar to concept 4, but with gills/louvers in the radial direction to create turbulence and draw in more air (ejector principle).

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total</td>
<td>Disc centre</td>
<td>Disc channels</td>
</tr>
<tr>
<td>-1.1%</td>
<td>-37%</td>
<td>-25%</td>
</tr>
</tbody>
</table>

Comment/Analysis:
No improvement compared to Concept 4.
### Concept 6

**Description/Idea:** Inlet through the three lower holes in the upright, air is fed both to the centre of the disc and along the inside disc surface.

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td></td>
<td>-0.3%</td>
<td>-8.0%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** Small improvement on inside disc surface and inside pad but worse in disc channels, the airflow is not sufficient to cool both inside surface/pad and channels.

(Brake disc removed)

### Concept 7

**Description/Idea:** Inlet in front of upright, the flow is distributed in a U-shaped channel along the inside disc surface and then the air is escaping through a 2.5 mm gap between duct and disc.

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td></td>
<td>1.9%</td>
<td>-32%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** Small overall improvement from baseline. Improvement on inside disc and pad but also worse on the centre/channels and outside.
After analysing the results at this stage it was apparent that the hose airflow was not sufficient to improve the cooling of both the disc channels and the external surface. It was easy to improve the heat transfer rate at a particular area but then it resulted in less cooling in another location. This is because the cooling effect from the surrounding airstream is quite strong. All duct designs that cover a significant part of the hot surfaces are also removing the cooling effect of the surrounding airstream from that area.

It was also concluded that letting the disc channels aspirate naturally (while only directing the hose airflow over the disc external surface) was not an effective route. The channels are the biggest contributor to the cooling which means that a great deal of heat transfer is lost if the channel flow is decreased (as was the case with natural aspiration).

The criterion regarding a hose attachment with small displacement during cornering was also difficult to fulfil with the current set of constraints. Only concept 6 featured a hose attachment position that fulfilled the criteria relatively well, but that design clashed with the wheel speed sensor and was not ideal. In the other designs the inlet is placed either side of the upright, which means rather big displacement during cornering. The hose attachment position could be moved closer to the wheel axis with some additional bends on the ducting, but that would inhibit the flow.

From concept 1 and 2 (in which all air is forced to the disc centre/channel inlets) it was learned that the direction of the air as it enter the centre has great influence on the channel flow rates. As there are small gaps between the centrepiece and the brake disc, some air can escape there. So even with these two designs, all hose flow is not forced through the channels. The hose airflow in this project is so much inertia driven and less pressure driven that it is important to avoid abrupt direction changes which inhibits the flow.

Both the hose attachment criteria and the need for a smooth radial flow path to the disc channels, led to the idea of an inlet in the centre of an axle with radial channels. This would fulfil the criteria regarding the hose attachment position well and the axle would also act like a centrifugal pump to some extent. As the idea of a different axle design was outside the project constraints, it had to be approved by Koenigsegg. The company response was that it was feasible if the results were good enough to motivate it. Koenigsegg also suggested a design with scrapers on the exits of the disc channels, this resulted in concept 9. That design also led to another idea, concept 10, which is designed to direct the channel exit flow along the outside disc surface.

It was also suggested to test a "passive" brake cooling design that is used by a competing brand. It consists of two ring-shaped plates, one on each side of the disc, with punched/flanged slots towards the disc.
### Concept 8

**Description/Idea:** Inlet duct in the centre of a modified axle with radial channels, the through hole is plugged. The brake disc centre is closed off so no air is escaping there.

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td></td>
<td>14%</td>
<td>34%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** Good overall improvement, better cooling on most surfaces. Axle channels and disc channels are working like a centrifugal pump and increases the hose flow rate which benefits the cooling.

### Concept 9

**Description/Idea:** Inlet through axle as in concept 8. Shield inboard of disc surface with angled scrapers at disc channel exits. The hose airflow is distributed between channels and the space between disc surface and shield. The scrapers at the disc outer diameter are supposed to convert the angular motion of the channel exit flow to a axial flow and increase the airflow through the rim.

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td></td>
<td>7.7%</td>
<td>-6.3%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** Less effective than Concept 8, the hose airflow is not sufficient for cooling both inside surface and centre/channels. Also this concept results in less cooling of the outside of the disc.
### Concept 10

**Description/Idea:**
Inlet through axle as in concept 8 with closed disc centre. Cylindrical shield close to the inside of the rim which is curved around the outer edge of the disc to direct flow over the outside disc surface. Angled vanes that convert the angular motion of the channel exit flow to a axial flow.

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total 4.6%</td>
<td>Disc centre -4.3%</td>
</tr>
<tr>
<td></td>
<td>Disc channels 28%</td>
<td>Disc outside -35%</td>
</tr>
<tr>
<td></td>
<td>Disc inside 8.7%</td>
<td>Pad inside 21%</td>
</tr>
<tr>
<td></td>
<td>Disc outside 11%</td>
<td>Pad outside 11%</td>
</tr>
<tr>
<td></td>
<td>Hose 52%</td>
<td>Channels 68%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** The cooling of the outside disc surface was improved, but the cooling of the inside was reduced more. In general it was worse than e.g. concept 8, even if the flow rates were higher.

### Concept 11

**Description/Idea:**
Inlet through axle with closed disc centre as in concept 8 but a portion of the flow is used to blow between the rim and caliper. Ring-shaped plates 8 mm from the disc inside and outside surfaces with flanged slots (5mm from disc).

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total 15%</td>
<td>Disc centre -5.2%</td>
</tr>
<tr>
<td></td>
<td>Disc channels 25%</td>
<td>Disc inside 15%</td>
</tr>
<tr>
<td></td>
<td>Disc outside 18%</td>
<td>Pad inside 21%</td>
</tr>
<tr>
<td></td>
<td>Pad outside 17%</td>
<td>Hose 73%</td>
</tr>
<tr>
<td></td>
<td>Channels 66%</td>
<td>Channels 66%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** Good overall improvement, strange that the cooling of the disc centre is worse than concept 8. The ring-shaped plates improve the convective cooling on both inside and outside disc surfaces.
### RESULTS

**Concept 12**

**Description/Idea:** Inlet through axle, chamfer added to radial channels inlets. Closed disc centre as in concept 8. Ring-shaped plates 8 mm from the disc inside and outside surfaces with flanged slots, similar to concept 11, but with inverted angle and deeper flanged slots (3.5 mm from disc).

<table>
<thead>
<tr>
<th>Difference from baseline</th>
<th>Heat transfer rate</th>
<th>Mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total</td>
<td>Disc centre</td>
</tr>
<tr>
<td>25%</td>
<td>-2.0%</td>
<td>28%</td>
</tr>
</tbody>
</table>

**Comment/Analysis:** Good overall improvement, very big improvement on disc outside compared to concept 11.

### 6.3 Analysis and discussion

It is quite clear that the concepts with flow through the axle are the most efficient designs in order to improve the cooling. The position of the inlet in these concepts is also best with regard to the criteria about minimum hose attachment displacement during cornering. Concept 8 is the most effective design with regard to the forced convection cooling and it involves few and simple additional parts.

The "passive" cooling solution with the ring-shaped plates on either side of the disc showed promise in the simulations, concept 11 showed about 15% improvement on both inside and outside. Concept 12 showed 78% (!) improvement on the outside. The way it probably works is that the plate introduces some shearing in the air between the plate and the moving surface of the disc and creates turbulence. The slots contribute to this and they also mixes in some fresh air. See Appendix 2 for comparison of flow plots between Concept 8 and Concept 12. The part of the hose flow that escapes between the disc and the disc centerpiece are also captured inside the outside plate. It can also be that the channel exit flows creates a wake/low pressure zone on either side of the disc which draws air radially outwards. It is however important to point out that only convection was considered in the...
RESULTS

Simulations, so especially the results relating to the passive cooling solution are not entirely valid. In reality the hot disc radiate heat, some of the radiation heat will be reflected back to the disc and some of the radiation will be absorbed by the plate. So the cooling effect of the plates are probably significantly lower than the results indicate, they could actually be counterproductive. In order to minimize the amount of heat reflected back to the disc, the plates should have a black surface coating. The effect of the ring-shaped plates should be tested experimentally, which should be relatively easy and not too costly to do. One benefit with the plates is that they protect the wheel from radiant heat, mostly in an axial direction.

Comparison: Baseline and Concept 8

Figure 6.5: Baseline velocity cut-plot, horizontal plane.

Figure 6.6: Concept 8 velocity cut-plot, horizontal plane.
RESULTS

Figure 6.7: Baseline velocity, cut-plot through disc channels.

Figure 6.8: Concept 8 velocity, cut-plot through disc channels.
6.4 Proposed solution
The proposed solution is concept 8, but the ring-shaped plates (concept 12) should be tested as they may improve the cooling further and they also protect the wheel spokes from radiant heat.

6.4.1 Inlet duct
The inlet duct and its holder can be made symmetric so the same part can be used on both sides of the car. It can be manufactured of carbon fibre composite, in two parts that are bonded together (see Figure 6.9). It can also be made of a fibre-reinforced plastic material using additive manufacturing technology. This design also protects the wheel speed sensor tone ring from contamination.

Figure 6.9: Inlet duct
6.4.2 Axle with radial channels

The axle design needs to be analysed with FEM in order to verify that the channels don't make the design too weak. It should however be possible to add some material where it is needed to increase the strength if necessary, without affecting the flow significantly. The axle design used in the simulations has an identical weight as the original part. The plug in the axle can be lathed from aluminium and secured with press fit or it can be incorporated in the axle part.

![Axle with radial channels](image)

Figure 6.10: Axle with radial channels

6.4.3 Centre lid

The lid that prevents the airflow from escaping from the centre of the disc needs to withstand high temperatures. So it is suitable to manufacture it from metal, for example from aluminium sheet metal. It is mounted to the axle with six screws as proposed below.

![Centre lid](image)

Figure 6.11: Centre lid
7 Discussion and conclusions

7.1 Conclusions from the project
It can be concluded that improving the cooling was more difficult than anticipated. This is probably because the natural cooling is very strong. The flow rate in the cooling hose is too low to make much difference. If the flow rate in the hose was increased with a fan then it would be possible to direct the flow over a bigger portion of the brakes. The meshing problems encountered may also contribute to less sensitive convective heat transfer. If the flow condition isn't solved correctly in the boundary layers then the effect of additional cooling flow may be smaller than it would be otherwise.

The results should be used with some caution, as the precision has not been validated. Especially, since there is some mesh dependency in the model. Other factors, such as turbulence modelling and boundary conditions, are also potential causes for inaccuracy.

7.2 Recommendations for further work
First and foremost should the mesh dependency be investigated and eliminated, this will however require a more powerful computer. It should be investigated if the MRF approach for rotating parts is suitable for this kind of case, it would enable the simulation to be run as steady state which is less demanding, but then another CFD software needs to be used. The turbulence modelling should be looked into in greater detail too improve the accuracy.

Regarding cooling solutions there is an endless number of variations to try.
8 Critical review

This project has a rather small impact on society in general and the risk that it causes any harm to anybody is small. However, if the brake cooling is not effective enough when driving on a race track, then there is a small risk that it could result in a crash and possibly injury. These kinds of developments are important as the knowledge can be applied to other more important matters. And it is a necessity for Swedish industry to be up to date with cutting edge technology in order to stay competitive on the global market. Which, in the long term means job opportunities and good welfare.
References


Appendix 1

CFD Configuration

General Info

<table>
<thead>
<tr>
<th>Units system</th>
<th>SI-modified</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis type</td>
<td>External (exclude internal spaces)</td>
</tr>
<tr>
<td>Exclude cavities without flow conditions</td>
<td>On</td>
</tr>
</tbody>
</table>

Computational Domain

Size

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>X min</td>
<td>-1.000 m</td>
</tr>
<tr>
<td>X max</td>
<td>0.600 m</td>
</tr>
<tr>
<td>Y min</td>
<td>-0.338 m</td>
</tr>
<tr>
<td>Y max</td>
<td>0.463 m</td>
</tr>
<tr>
<td>Z min</td>
<td>-0.100 m</td>
</tr>
<tr>
<td>Z max</td>
<td>0.601 m</td>
</tr>
</tbody>
</table>

Boundary Conditions

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2D plane flow</td>
<td>None</td>
</tr>
<tr>
<td>At X min</td>
<td>Default</td>
</tr>
<tr>
<td>At X max</td>
<td>Default</td>
</tr>
<tr>
<td>At Y min</td>
<td>Default</td>
</tr>
<tr>
<td>At Y max</td>
<td>Default</td>
</tr>
<tr>
<td>At Z min</td>
<td>Default</td>
</tr>
<tr>
<td>At Z max</td>
<td>Default</td>
</tr>
</tbody>
</table>

Physical Features

Heat conduction in solids: Off
Time dependent: On
Gravitational effects: Off
Rotation: Local region(s) (Sliding)
Flow type: Laminar and turbulent
High Mach number flow: Off
Humidity: Off
Default roughness: 50.0 micrometer
Default wall conditions: Adiabatic wall
### Ambient Conditions

| Thermodynamic parameters | Static Pressure: 101325.00 Pa  
Temperature: 20.00 °C |
|--------------------------|-----------------------------|
| Velocity parameters     | Velocity vector  
Velocity in X direction: -41.700 m/s  
Velocity in Y direction: 0 m/s  
Velocity in Z direction: 0 m/s |
| Turbulence parameters   | Turbulence model: Modified k-ε  
Turbulent energy: 1 J/kg  
Turbulent dissipation rate: 1 W/kg |

### Rotating regions

<table>
<thead>
<tr>
<th>Component</th>
<th>Rotating_region_disc</th>
<th>123.500 rad/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotating Region Rim</td>
<td>Component</td>
<td>Rotating Region Rim</td>
</tr>
</tbody>
</table>

### Boundary Conditions

| Type                     | Total pressure  
Total Pressure: 103900.00 Pa  
Temperature: 20.00 °C |
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Disc Rotating Real Wall with Temp</td>
<td>Type</td>
</tr>
<tr>
<td>Roughness</td>
<td>50.0 micrometer</td>
</tr>
<tr>
<td>Wall temperature</td>
<td>600.00 °C</td>
</tr>
</tbody>
</table>
| Moving wall parameters   | Translation velocity: 0 m/s  
Angular velocity: 123.500 rad/s  
Direction: Axis of coordinate system  
Axis: Z |
### Disc Real Wall with Temp

<table>
<thead>
<tr>
<th>Type</th>
<th>Real wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roughness</td>
<td>100.0 micrometer</td>
</tr>
<tr>
<td>Wall temperature</td>
<td>500.0 °C</td>
</tr>
</tbody>
</table>

### Pads Real Wall with Temp

<table>
<thead>
<tr>
<th>Type</th>
<th>Real wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roughness</td>
<td>50.0 micrometer</td>
</tr>
<tr>
<td>Wall temperature</td>
<td>450.0 °C</td>
</tr>
</tbody>
</table>

### Inlet Mass Flow Radiator

<table>
<thead>
<tr>
<th>Type</th>
<th>Inlet Mass Flow</th>
</tr>
</thead>
</table>
| Flow parameters       | Flow vectors direction: 3D vector  
                        | Mass flow rate: 1.0000 kg/s  
                        | Relative to rotating frame: No  
                        | Relative component in X direction: 2.0000  
                        | Relative component in Y direction: -1.0000  
                        | Relative component in Z direction: 0  
| Thermodynamic parameters | Approximate pressure: 101325.00 Pa  
                           | Temperature: 30.00 °C          |
| Turbulence parameters | Boundary layer type: Turbulent |
| Boundary layer parameters |                       |

### Real Wall Ground

<table>
<thead>
<tr>
<th>Type</th>
<th>Real wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roughness</td>
<td>1000.0 micrometer</td>
</tr>
</tbody>
</table>
| Moving wall parameters | Translation velocity: -41.700 m/s  
                        | Angular velocity: 0 rad/s  
                        | Direction: Axis of coordinate system  
                        | Axis: Y               |

### Real Wall Wheel Rotation

<table>
<thead>
<tr>
<th>Type</th>
<th>Real wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roughness</td>
<td>50.0 micrometer</td>
</tr>
</tbody>
</table>
| Moving wall parameters | Translation velocity: 0 m/s  
                        | Angular velocity: 123.500 rad/s  
                        | Direction: Axis of coordinate system  
                        | Axis: Z               |
Calculation Control Options

Finish Conditions

<table>
<thead>
<tr>
<th>Finish Conditions</th>
<th>If all are satisfied</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum iterations</td>
<td>270</td>
</tr>
<tr>
<td>Goals convergence</td>
<td>Analysis interval: 5</td>
</tr>
</tbody>
</table>

Solver Refinement

Refinement: Disabled

Advanced Control Options

Flow Freezing

<table>
<thead>
<tr>
<th>Flow freezing strategy</th>
<th>Disabled</th>
</tr>
</thead>
</table>

Manual time step (Freezing): Off
Manual time step: 1.413e-004 s

Material Settings

Air

Path: Gases Pre-Defined
Specific heat ratio (Cp/Cv): 1.399
Molecular mass: 0.0290 kg/mol
Dynamic viscosity

![](chart.png)
Specific heat (Cp)

Thermal conductivity

![Graph of Specific heat (Cp)](image1)

![Graph of Thermal conductivity](image2)
Appendix 2

Comparison: Concept 8 and 12

Figure A2.1: Concept 8, velocity 1 mm inside the disc.

Figure A2.2: Concept 8, velocity 1 mm outside the disc.
Figure A2.3: Concept 12, velocity 1 mm inside disc.

Figure A2.4: Concept 12, velocity 1 mm outside disc.
Figure A2.5: Concept 8, temperature horizontal plane

Figure A2.6: Concept 12, temperature horizontal plane