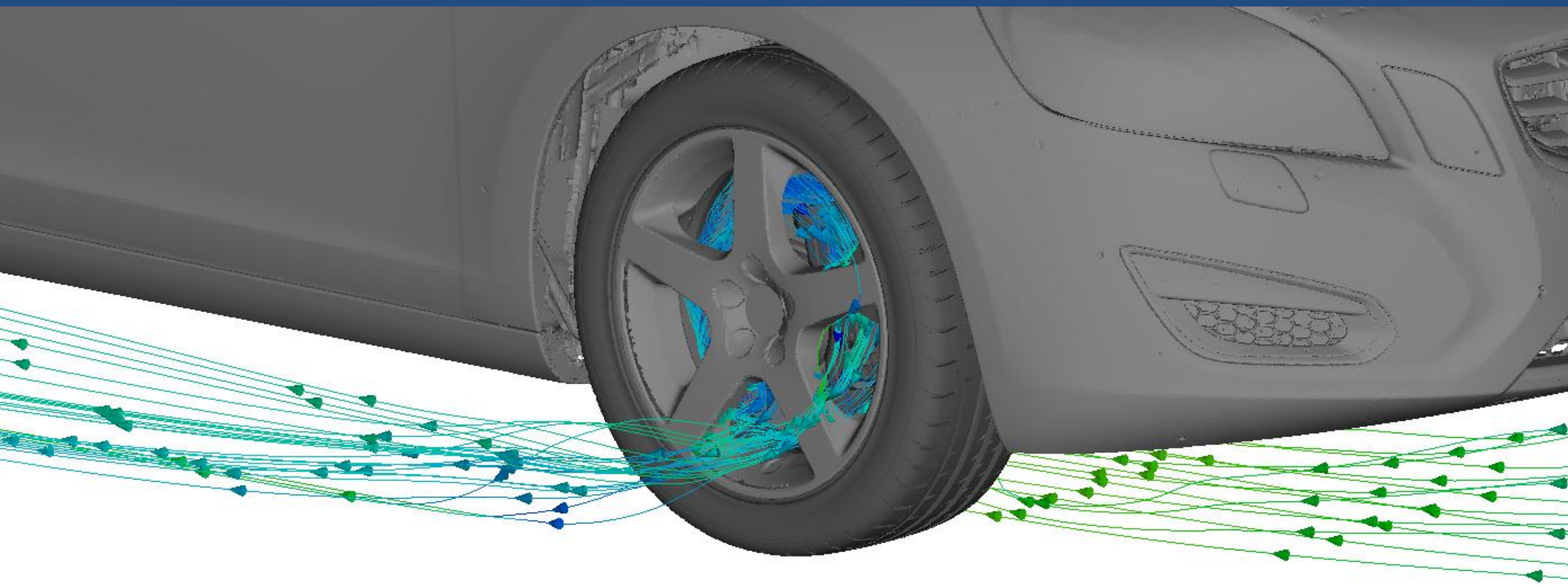
CHALMERS



CFD and Design Analysis of Brake Disc

Master's Thesis in Automotive Engineering

ADRIAN THURESSON

Department of Applied Mechanics
Division of Vehicle Engineering and Autonomous Systems
Road Vehicle Aerodynamics and Thermal Management
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2014

Master's Thesis 2014:11

MASTER'S THESIS IN AUTOMOTIVE ENGINEERING

REPORT NO. 2014:11

CFD and Design Analysis of Brake Disc

ADRIAN THURESSON

Department of Applied Mechanics
Division of Vehicle Engineering and Autonomous Systems
Road Vehicle Aerodynamics and Thermal Management
CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2014

CFD and design analysis of brake discs
ADRIAN THURESSON

Master's Thesis 2014:
ISSN 1652-8557
Department of Applied Mechanics
Division of Vehicle Engineering and Autonomous Systems
Road Vehicle Aerodynamics and Thermal Management
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:
The cover figure describes the wheel house flow behaviour of a mid-size Volvo model driving in 100 km/h with a disc temperature of 200°C. The plotting tool is a particle trace application in Ensight 10.0 where the lines of colour represent the velocity magnitude and direction.

Gothenburg, Sweden 2014

CFD and design analysis of brake disc
Master's Thesis in the *Automotive Engineering*
ADRIAN THURESSON
Department of Applied Mechanics
Division of Vehicle Engineering and Autonomous Systems
Road Vehicle Aerodynamics and Thermal Management
Chalmers University of Technology

ABSTRACT

The ever increasing need of effective transportations puts automobile manufacturers in a non-avoidable situation of maintaining and improvement of safety systems. The brake system has always been one of the most critical active safety systems. Brake cooling is further an important aspect to consider for brake disc durability and performance.

The importance of convective cooling of a brake disc is an important factor since it can be significantly improved by trivial design changes and contributes to the major part of the total dissipated heat flux for normal driving conditions.

At first, experimental aero-thermal flow test of ventilated brake discs were performed in the Volvo brake machine and the Volvo wind tunnel. These tests were further correlated with a CFD modelling method in order to provide reliability and trust to the model itself. The developed CFD model was then applied into a full vehicle model of a mid-size Volvo to predict and analyse the aero-thermal flow behaviour of a ventilated brake disc in the wheel house.

It can be seen that the vanes contributes to the major dissipated convective heat from the brake disc and the wheel house through-flow is of great importance for the cooling behaviour. Because of the complexity of the wheel house flow, trivial design changes could alteration the aero-thermal flow behaviour of the brake disc. An even more comprehensive study could be obtained by applying a full thermal which predicts the aero-thermal flow dependency of convection, conduction and radiation.

Keywords: Brake Cooling, Automotive Design, CFD Model, Turbulence Model, Pump Flow, Wind Tunnel Experiment, Brake Test Machine, Brake Shield, Convective Heat Transfer, Wheel House Design.

CFD och design analys av bromsskiva
Examensarbete inom Fordonsteknik
ADRIAN THURESSON
Institutionen för tillämpad mekanik
Avdelningen för fordonsteknik och autonoma system
Aerodynamik och Termodynamik
Chalmers tekniska högskola

SAMMANFATTNING

Det ständigt ökande behovet av effektiva transporter sätter biltillverkare i en oundviklig situation till att leverera säkrare, billigare och effektivare bilar. För att bibehålla och förbättra säkerhetssystemen för att anpassa sig till de ständigt ökande säkerhetskraven utan att öka fordonets vikt, utvecklas idag fler och fler aktiva säkerhetssystem. Dock har bromssystemet alltid varit en av de mest kritiska delarna när krav som dessa måste uppfyllas. Bromskylning är en viktig aspekt att beakta för bromsskivans hållbarhet och prestanda.

Vikten av konvektiv kylning av en bromsskiva är en viktig faktor eftersom den bidrar till den största delen av den totala, avgivna värmen för normala körförhållanden och kan förbättras avsevärt genom triviala designförändringar.

Inledningsvis så utfördes experimentella flödestester av ventilerade bromsskivor i Volvos bromsmaskin och Volvos vindtunnel. Dessa tester korrelerades senare med CFD-modelleringsmetoder för att ge tillförlitlighet till modellen. Den utvecklade CFD-modellen applicerades därefter i en fullskalig medelstor Volvo modell för att förutsäga och analysera konvektiva flödesbeteenden i bromsskiva och hjulhus.

Det kan ses att flödeskanalerna bidrar till den största avledda konvektionsvärme-delen från bromsskivan och genomflödet i hjulhuset är av stor betydelse för den kylande beteende. På grund av hjulhusflödets komplexitet kunde triviala designförändringar förbättra det termiska flödesbeteende hos bromsskivan. En ännu mer omfattande studie skulle kunna uppnås genom att tillämpa en fullständig termisk modell som förutsäger flödeberoende av konvektion, värmeledning och strålning.

Nyckelord: Bromskylning, Fordonsdesign, CFD modell, Turbulens Model, Pumpflöde, Vindtunnel Experiment, Bromstest-maskin, Bromssköld, konvektiv värmeöverföring, Hjulhusdesign.

Acknowledgements

The brake performance department at Volvo Cars has been very welcoming and helpful throughout both my summer work and my thesis project. With their comprehensive expertise and knowledge of automotive systems, they have continuously been providing direct and great inputs through uncertainties and problems. Few departments are filled with such cohesion and competence as this.

Special thanks go to my supervisor Daniel Gunnarsson who has been involved in most of what I've been doing during my time at VCC. With his curiosity, expertise and drive for CAE development, we've manage to create a great team that never saw any unsolvable problems, but only solution opportunities. Without his interest, engineering skills and management of the subject, the project would never have reached the results as it did.

The CFD/Aerodynamics department also deserves many thanks for their involvement in the project and its issues and development. Special thanks to Carl Andersson who has continuously provided with his impressive CFD expertise and automotive knowledge.

I would also like to thank my examiner, Lennart Löfdahl and my Chalmers supervisor, Simone Sebben for their guidance and advices during the project.

To sum up, my thesis project has been widely scoped among different departments and areas, so there are many people to thank for making this project possible. Some of them are Jörgen Hansson and the guys at PV16 for their involvement in preparation and tests, Mikael Mellander and the guys at PVT for the help with wind tunnel preparation and expertise, and the guys at PV14, PVKA and the metal workshop for their help with test equipment, material supply, manufacturing tools and construction work.

At last I would like to thank my friends and family for all support and encouragement throughout the project and also during my years as a student at Chalmers.

Adrian Thuresson

List of symbols

Acronym

Abbreviation	Description
VCC	Volvo Car Corporation
CAE	Computer Aided Engineering
CFD	Computational Fluid Dynamics
MRF	Multiple Reference Frame
MCI	Mesh Convergence Index
Re	Reynolds number
rpm	Revolutions Per Minute
CPU	Central Processing Unit
BC	Boundary-Condition
SS	Steady-State
VWT	Volvo Wind tunnel
CAD	Computer Aided Design
Vane/Vent	Ventilations channel of brake disc
Vent-in	Inlet of ventilations channels are directed inwards the vehicle
Vent-out	Inlet of ventilations channels are directed outwards the vehicle

Roman upper case letters

Symbol	Unit	Description
C_p	J/℃ kg	Specific heat per unit mass
T	℃ / K	Temperature in Celsius or Kelvin
$\dot{Q}$	J/s	Heat rate
Q	J	Heat amount
V	km/h	Velocity
A	m^2	Area
$\vec{J}$	J	Diffusion flux
R	-	Universal gas constant
M	g/mol	Molecular weight
S	-	Sutherland constant
J	kg$\cdot m^2$	Rotational inertia

Roman lower case letters

Symbol	Unit	Description
m	kg	Mass
t	s	Time
h	W/m^2 K	Convective heat transfer coefficient
v	m/s	Velocity

$\dot{m}$	kg/s	Mass flow rate
l	m	Length
ρ	kg/m^3	Density
λ	S/m	Thermal conductivity
σ_b	-	Stefan-Boltzmann constant
ξ	$W \cdot s^{1/2} \cdot K^{-1} \cdot m^{-2}$	Thermal effusivity
σ	-	Heat partition coefficient
ω	$rad \cdot s^{-1}$	Angular velocity
$\dot{\omega}$	$rad \cdot s^{-2}$	Angular acceleration
ε	-	Emissivity

Sub-scripts

Notification	Description
disc/d	Brake disc
pin	Pin design-version of brake disc
vane	Straight ventilation channels design-version of brake disc
pads/p	Brake pads
in	Inlet
out	Outlet
atm	Atmospheric
cond	Conduction
conv	Convection
rad	Radiation
r	Radial direction
t	Tangential direction
carr	Carrier
avg	Average
∞	Ambient
0	Initial state/reference value
eff	Effective
op	Operating
const	Constant
IG	Ideal gas approach
IIG	Incompressible ideal gas approach
ss	Steady-State
dyn	Dynamic
stat	Static
env	Environment/surroundings
obj	Object
w	Wheel
fric	Friction
pt	Pad torque

Table of Content

1 INTRODUCTION

In this chapter a comprehensive introduction to the project in form of background, history, fundamental brake knowledge and project motivation will be presented.

1.1 Background

The ever increasing need of effective transportations puts automobile manufacturers in a non-avoidable situation of delivering safer, cheaper and more effective cars. For a car manufacturer, it is necessary to keep up with competitors within these areas in order to maintain and attract consumers. According to ASIRT (2013), nearly 1.3 million people worldwide die in traffic accidents every year, which makes a rate of 3,287 deaths each day, and approximately 30 million people are disabled or injured, mostly younger people between 15 and 44 years of age. Due to statistics like this, the need for safer cars today becomes more and more important and the requirements of safety systems of the cars even tougher.

Volvo Car Corporation (VCC) is one of the strongest automobile industry brands with a very long and proud history of state of the art, world leading innovations. Today, VCC is one of the very leading car manufacturers in the world considering traffic safety engineering and has long been stressed and marketed for their historical reputation of solidity, reliability and safety.

To maintain and improve safety systems in order to adapt to the ever toughening safety requirements without increasing vehicle weight, more and more active safety systems are today developed. However, the brake system has always been one of the most critical parts when requirements like these have to be met.

1.2 Brake system description

In most of the modern cars today, disc brakes are used on the front wheels and in most cases also on the rear wheels. The main purpose of a disc brake system is to decelerate the vehicle by transforming the kinetic energy of the car into thermal energy by friction between the brake disc and the brake pads.

The driver decides when to decelerate the vehicle by pushing the brake pedal which determines the brake fluid pressure inside the hydraulic circuit. To increase the hydraulic pressure higher than the force applied from the driver, a booster is used which uses vacuum. For gasoline engines, this vacuum usually comes from the vacuum that occurs in the intake manifold of the combustion engine. For diesel engines, a separate vacuum pump is instead often used. This amplified force that comes out of the booster goes into the master cylinder which distributes the pressure out to each caliper.

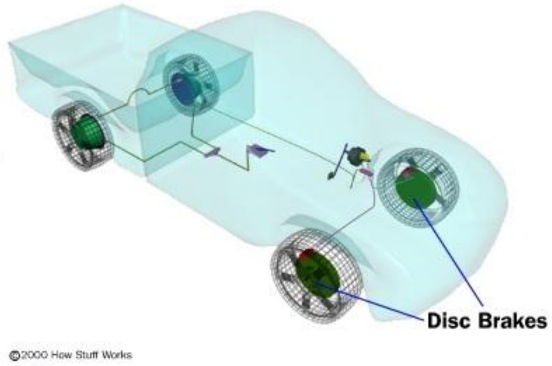


Figure 1.1: Disc brake system location in a car, taken from Howstuffworks (2000)

Single-piston floating caliper is the most common disc brake type in cars, but all the way up to six pistons is used in some braking systems today, i.e. three pistons on each side of a fixed caliper braking system.

1.2.1 Fixed caliper

A disc brake system which uses fixed coupled pistons, e.g. one, two or three pistons on each side, is called a fixed caliper braking system. Pistons on both sides are pushing respective brake pad directly against respective side of the brake disc, to create a frictional force which provides a braking momentum on the rotor.

1.2.2 Floating caliper

A floating caliper braking system uses pistons on one side of the brake disc only. These pistons push the brake pad against the inner side of the rotor at the same time as they push the caliper backwards with the same force. The resultant force of the caliper affects the other side of the rotor where the outer brake pad will push the outer rotor side.

As mentioned earlier in the chapter, the most common disc brake type is with a floating caliper braking system. The floating caliper is often preferred over fixed caliper due to both economical and mass/volume motivations. The cost will increase due to the double amount of pistons and the system will take up more volume and mass in the wheelhouse with a fixed caliper system.

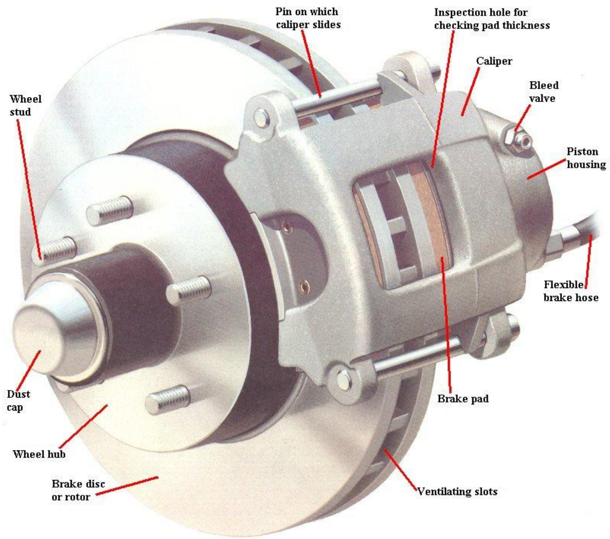


Figure 1.2: Floating caliper disc brake system in detail, taken from Motorera (2013)

The main components in a disc brake are the brake pads, the caliper and the brake disc/rotor. These are visualized above in figure 1.2.

1.2.3 Brake disc design

Brake discs today are shaped in many different ways. The most common front brake discs designs are vane designs, see left picture in figure 1.3 below, and pin designs, see right picture in figure 1.3 below, taken from Pulugundla (2008).

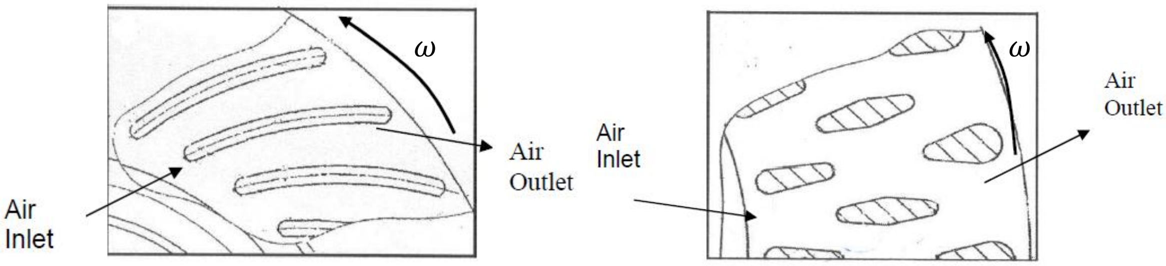


Figure 1.3: Sketch of vane design (left) and pin design (right) of ventilation channels.

The vane design usually provides a more stiff structure and better cooling performance while the pin design usually is preferred in sound point of views. The focus in this study will be on straight vane designs with both a vent-in and a vent-out solution. The limitation of only investigating straight vane design comes from the future design directions of VCC and the time wouldn't be enough to investigate another. The pin design is though physically investigated for the aero-thermal behaviour during the brake machine tests, to gain some informational input of other design concepts. When talking about vent-in brake disc designs, it means that the inlet of the ventilation channels is directed towards the inside of the car seen from a wheel house point of view, see left side of figure 1.4 below. The vent-out brake disc design is the opposite way, the inlet of the ventilation channels is directed outwards seen from a wheel house point of view, see right side of figure 1.4 below.

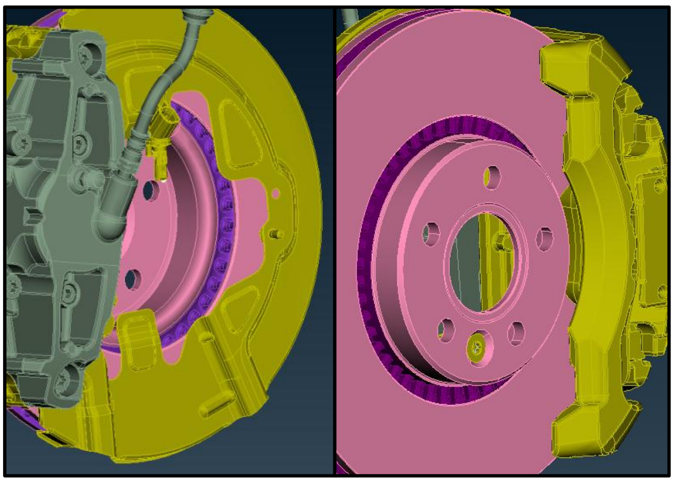


Figure 1.4: Vane-in design to the left seen from inside of the vehicle and vane-out design to the right seen from the outside of the vehicle.

The major advantage of vent-out designs is the preferred stiffness and displacement during heating in the brake disc structure, which then becomes easier to achieve. For vent-in designs, the outside of the brake disc, the side seen from viewers, is easier to pre-treating to prevent corrosion when the ventilation inlet is on the other side. From a design point of view, the vent-in therefore is preferred. Which one of the two that is preferred in a cooling point of view, remains to be investigated later on in chapter 7. A cross-section view of a vent-in (to the left) and a vent-out (to the rights) can be seen in figure 1.5 below.

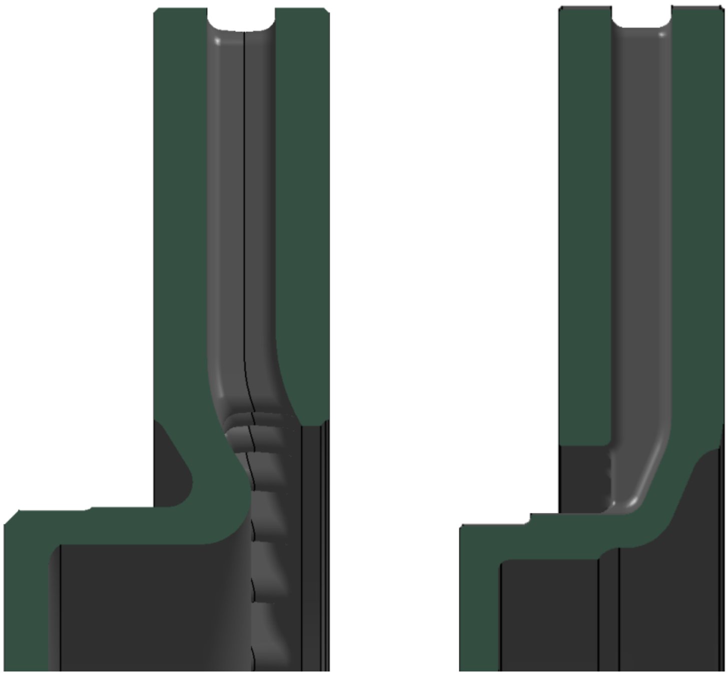


Figure 1.5: Cross section view of a vane-in and a vane-out ventilations channel design.

1.2.4 Brake shield

Almost all car manufacturers today are using brake shield today, they're placed on the inside of the brake disc, see figure 1.6 below. Its main purpose is to protect the brake disc from all dirt and splash than comes up with the flow underneath the car. These dirt particles will interfere with the contact patch between the brake disc and the brake pads and will further decrease the brake friction force. The unwanted dirt will also damage both the brake disc and the brake pads which will decrease the life time of the products.

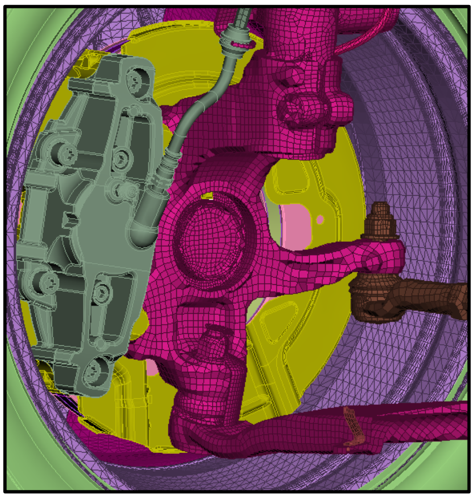


Figure 1.6: Brake shield (yellow plate) seen in a wheel house view from inside, underneath a mid-size Volvo model.

There are not only advantages with brake shield though, as most of automotive design developments, even this is a trade-off. It has been known that brake shields can generate noise. Mostly the brake shield covers the brake disc from road dirt but sometimes small stones and other unwanted products get stuck between the brake shield and the brake disc which further creates a scratching and rattling noise which make the driver and passenger uncomfortable and the customer thinks it’s something wrong with the brake. Second disadvantage is what this project will partly investigate, the blocked cooling flow that appears when the shield covers too much of the brake disc. This will later on be studied in chapter 7; design analysis.

1.3 Brake disc failure modes

To understand the importance of a comprehensive design investigation of a brake disc, a deeper understanding of the different thermal failure modes is necessary. An example of an overheated brake disc can be seen below in Figure 1.7.

Figure 1.7: Dynamometer test, overheated brake disc, taken from Eggleston (2000)

When non-uniform contact forces and/or overheating occur between the brake disc and the brake pads, so-called judder appears. Thermal judder, unlike cold judder, principally occurs as an effect of thermal instabilities in the brake disc material, often due to poor brake disc design. Examples of geometrical deflection effects like butterfly, coning and corrugated effects due to thermal judder can be seen below in figure 1.8, 1.9 and 1.10.

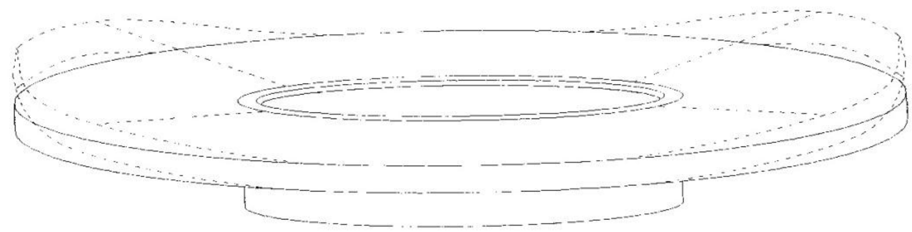


Figure 1.8: Butterfly effect due to thermal judder, taken from Eggleston (2000)

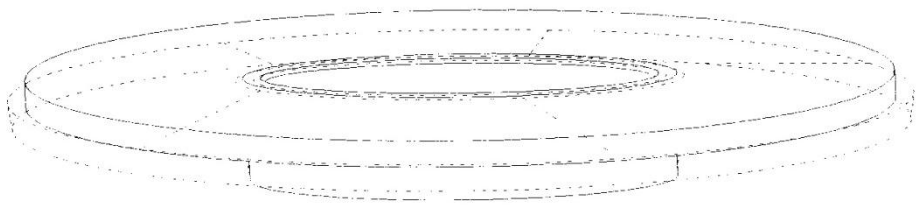


Figure 1.9. Coning effect due to thermal judder, taken from Eggleston (2000)

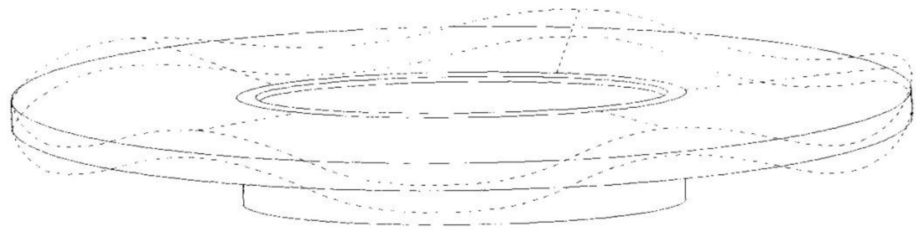


Figure 1.10: Corrugated effect due to thermal judder, taken from Eggleston (2000)

Cracking can also appear due to non-uniform heat distribution in the brake disc material. When non-uniform temperature distribution occurs, the brake disc will expand non-uniformly and therefore create stress concentrations and crack propagation might occur and damage the disc, this phenomenon is called hot spots. The most common solutions to avoid these hot spots are to redesign the brake disc to maximize heat dissipation and to make the temperature distribution more uniform.

1.4 Project motivation

As explained previous in the introducing chapter, most of the complications and failures with a brake disc is due to problems with heat distribution and heat level absorbed by the rotor. This absorbed energy in form of heat is dissipated through convection, conduction and radiation. The easiest of these dissipation areas to affect the impact of is convection. To be able to start modifying existing designs and constructions for better convective cooling performance, more knowledge of the flow behaviour has to be clarified. This project will therefore focus on cooling affected flow behaviour which exist in a mid-size Volvo model and if it can be simplified for a more effective and user-friendly evaluation tool.

1.5 Objectives and aims

The purpose of this study is to provide a comprehensive knowledge of the flow behaviour inside and around a brake disc in specific pre-defined cases and how these behaviours affect the cooling performance, e.g. the aero-thermal flow behaviour of the brake disc. Another aim of the project is to generate an overview of the possibilities of simplify the found aero-thermal flow behaviour and also investigate the design possibilities, both brake disc design focus and car body design focus, and their impact.

1.6 Limitations

For complex flow behaviour cases like these, it's important to be clear of the set limitations in the beginning in order to provide a consistent comparison later on. This study will only be performed with a specific mid-sized Volvo model with a five spoke rim design and standard wheel house equipment which should be remembered for the final conclusion. Any full thermal models will not be analyzed, only the convective flow behaviour will be studied for the different cases. The brake disc temperature will also be assumed to be constant over the surface since its otherwise dependent on the brake load case which is preferred not to have as an extra parameter.

Although future limitations will continuously be explained for the specific cases such as temperatures, velocities and designs.

Another limitation is that only the front brakes will be investigated in this study because they're normally prioritized due to the brake torque distribution higher in front which normally provides a lot of absorbed heat energy.

Also only steady-state thermal analyses will be used in this study, i.e. no transient cases. Industrial engineers often perform SS simulations like these before the transient analyses are performed to provide initial values of e.g. temperatures, thermal gradients and heat flow rates. Since most material properties are dependent on temperature, the analysis is non-linear and would therefore affect temperature dependent convective coefficient effects.

All limitations in this study provide a narrow range of applicable areas of the final result, but the result will provide a reliable and stable base for future extensive investigations.

1.7 Framing of questions

At last, the framing of questions needs to be defined to continuously have clear and concrete goals and directions with the project to avoid crossing project limitations and keep the focus on the right area. The first questions of issue for the project is based on a general theoretical issue to get a clear view of the continuing investigation followed by the three main examining questions which will be the key part of the results. The questions are mostly based on my personal interest but also towards the requests from VCC brake performance department.

- What are the basics of improved thermal management of a ventilated brake disc?
- How can the internal and external aero-thermal flow behaviour of a ventilated brake disc be simulated and analysed?
- How can a simplified evaluation model or method of a brake disc be developed and used, that represents a full vehicle aero-thermal flow behaviour?
- What are the design directions considering improvement of the aero-thermal cooling performances of a brake disc?

2 Theoretical behaviour

In order to provide a reliable and comprehensive investigation of the different brake disc behaviour, an introducing theoretical part is mandatory. This chapter will therefore take up the basic theoretical properties of a brake disc.

2.1 Formed heat management

In previous chapter 1, the main function of a brake disc system was explained, to reduce the kinetic energy of the vehicle by partially translating the energy into heat. The energy transformation is mainly made through the friction contact between the brake pads and the brake discs. When the brake pads are applied onto the brake disc, the pads will generate a brake force onto the brake disc which further generates a resisting torque on the wheel. The resisting opposite torque of the wheel torque comes from the ground by friction between the tyre and ground contact patch which further will decrease the vehicle speed. First, elastic energy is built up in the tyre due to elastic deformation to build up the friction force of the contact patch.

Not all absorbed heat energy is directly due to deceleration of the vehicle in the very start of braking, first elastic deformation of the chassis and suspension take place which absorbs energy. Also if the brake force is to large, slip will appear and there also generate heat energy instead of decreasing the kinetic energy. A synthesis figure of this torque balance can be seen in figure 2.1 below.

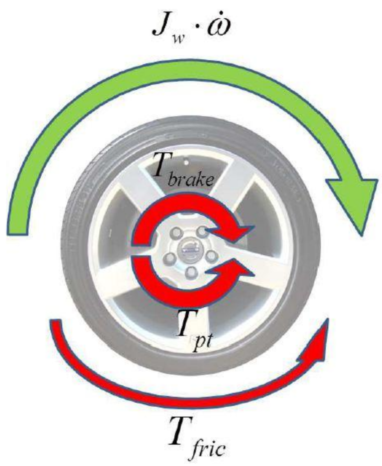


Figure 2.1: Torque balance overview of front right wheel, taken from Neys (2012).

Figure 2.1 above with related equation 2.1 below describe the dynamic evolution of a front right wheel; the pads apply a braking torque onto the brake disc (T_{pt}) which respond with an opposite direction torque (T_{brake}) while the ground friction torque (T_{fric}) opposes the inertia.

$$J_w \cdot \dot{\omega} = T_{brake} - T_{fric} - T_{pt} \qquad \text{Eq. 2.1}$$

Back to the main phenomena of the dynamic behaviour of a braking sequence, the heat created by a contact patch, in this case the pad-brake disc contact patch transformation into heat energy is of interest, is given by equation 2.2, 2.3 and 2.4 below, taken from Neys (2012).

$$\dot{Q} = F_{fric} \cdot v_{slide} \qquad \text{Eq. 2.2}$$

Or:

$$\dot{Q} = T_{brake} \cdot \omega \qquad \text{Eq. 2.3}$$

Where:

$$T_{brake} = F_{fric} \cdot r_{eff} \qquad \text{Eq. 2.4}$$

2.1.1 Absorbed disc heat

When it now exist a base on the theory of generated brake heat, the absorbed brake disc heat theory can be introduced. Not all generated heat is in reality distributed among the brake disc and the brake pads. According to Neys (2012), approximately 99% of the generated heat is distributed among the brake parts and the rest is directly dissipated into the surrounding air by means of convection. This fraction of absorbed brake disc heat is dependent of the overall conditions and design parameters, although Neys (2012) presents a theoretical model for a heat fraction coefficient between the brake pads and the brake disc. For imperfect contact between them, the partition coefficient for the brake disc can be calculated according to equation 2.5 below.

$$\sigma_d = \frac{\xi_d \cdot S_d}{\xi_d \cdot S_d + \xi_p \cdot S_p} \qquad \text{Eq. 2.5}$$

Where the fraction of the individual product of thermal effusivity (ξ) and contact surface (S) for each part represent the partition coefficient. The brake disc effusivity is there before calculated according to equation 2.X below, where λ is the thermal conductivity and $C_p$is the thermal capacity.

$$\xi_d = \sqrt{\lambda_d \cdot \rho_d \cdot C_{p,d}} \qquad \text{Eq. 2.6}$$

When a normal brake pad and disc combination was applied into equation 2.5 above with respectively material properties, it was found that the 98.8% of the total generated heat from a normal brake scenario was absorbed by the brake disc.

Initially, it was described that the brake disc performance is dependent on temperature of the disc lumped mass, which is further dependent on the net heat rate in and out of the system, i.e. the brake disc surface. This relationship can be described by equation 2.7 below, taken from Neys (2012). The lower the net heat rate is the lower the temperature gradient will be, since the mass is constant and the changes of C_p can be assumed negligible for small temperature changes, which will benefit the overall lifetime and performance of the brake disc. Since this study doesn't include the investigation of the brake disc heat input and how to lower/distribute the in-heat more efficient, the focus will instead be held at the out-heat management, and how to increase that factor in order to decrease the positive temperature gradient or increase the negative temperature gradient.

$$C_p \cdot m_{disc} \cdot \frac{dT}{dt} = \dot{Q}_{in,disc} - \dot{Q}_{out,disc} = \dot{Q}_{net,disc} \qquad \text{Eq. 2.7}$$

Where $\dot{Q}_{in,disc}$ is generated through the brake torque as described earlier with respect to the associated partition coefficient, see equation 2.8 below. The heat will thereafter distribute along the brake disc mass through conductive heat transfer.

$$\dot{Q}_{in,disc} = \sigma_d \cdot T_{brake} \cdot \omega \qquad \text{Eq. 2.8}$$

$$\dot{Q}_{out,disc} = \dot{Q}_{cond,disc} + \dot{Q}_{conv,disc} + \dot{Q}_{rad,disc} \qquad \text{Eq. 2.9}$$

The dissipated heat appears due to three main heat transfer modes; conduction, convection and radiation as described in equation 2.9 above, the dissipative heat management will be handled in the next chapter.

2.2 Dissipative heat management

The heat transfer from the brake disc to the ambient air and other wheel house parts takes place in three different heat transfer modes; conduction, convection and radiation, see figure 2.2.

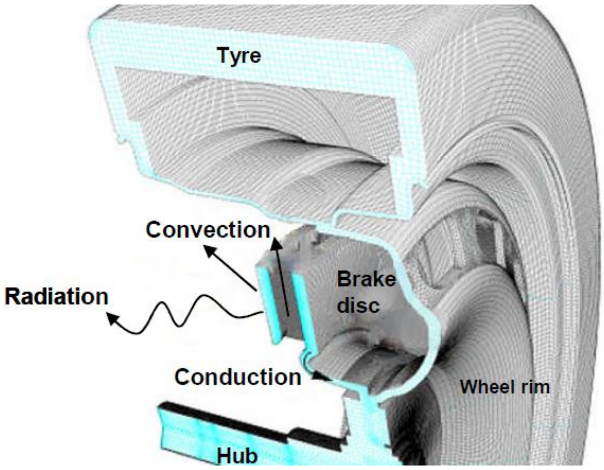


Figure 2.2: Heat transfer modes, taken from Pulugundla (2008)

2.2.1 Conduction

Thermal conduction or heat conduction is usually described as the energy transfer of heat in a solid between particles. Thermal conductivity is used as the material property for conducting (transfer) heat and is usually defined as λ. That means a material with a higher thermal conductivity transfers heat at a higher rate across the material compared to another material with lower thermal conductivity, see equation 2.10 and figure 2.3 below.

$$\dot{Q}_{cond} = \frac{A \cdot \lambda \cdot (T_1 - T_2)}{\Delta x} = A \cdot \lambda \cdot \frac{\Delta T}{\Delta x} \qquad \text{Eq. 2.10}$$

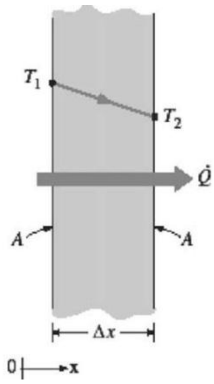


Figure 2.3: Visualization of the theory of thermal conduction

2.2.2 Convection

Convection is often referred as convective heat transfer which is the physical behaviour of heat transfer by moving fluids that transports heat energy from one place to another. This heat loss phenomena is also the main contributor of the total heat loss while driving and according to Newton, be explained as equation 2.11 below, see Incropera (2001).

$$\dot{Q}_{conv} = h_{avg} \cdot A \cdot (T_1 - T_2) = h \cdot A \cdot \Delta T \qquad \text{Eq. 2.11}$$

Where:

$$h_{avg} = \frac{1}{A} \int h \cdot dA \qquad \text{Eq. 2.12}$$

As can be seen in the definition of convective heat transfer rate is the dependency of the convective heat transfer coefficient, h, the area and the temperature difference. According to Thermopedia (2014), the convective heat coefficient is mainly dependent on the air flow around the brake disc, i.e. the flow velocity which generally increases the heat transfer rate, the turbulence intensity which usually increasing the heat transfer rate through increased intensity, and the flow structure.

2.2.3 Radiation

Another contributor to the total heat loss of the brake disc is the radiation heat transfer. Radiation is explained by the Stefan-Boltzmann law as given in equation 2.13 below.

$$\dot{Q}_{rad} = \varepsilon \cdot \sigma_b \cdot A \cdot \left(T_{obj}^4 - T_{env}^4\right) \qquad \text{Eq. 2.13}$$

Where ε is the emissivity ($0 \leq \varepsilon \leq 1$), σ_b is Stefan Boltzmann's constant ($5.67 \cdot 10^{-8}$), A is the surface area, T_{obj} is the temperature of the object and T_{env} is the temperature of the surroundings.

2.3 Mass flow rate

One on the most important factors to consider when developing a brake disc design for better internal cooling performance is the mass flow rate through the brake disc passages. When a brake disc is heated due to braking, it should cool down as fast as possible to avoid overheating (see 1.3 Brake disc failure modes). A higher mass flow rate through the rotor passage will in this case provide a higher cooling rate.

A disc brake rotor could therefore be seen as a centrifugal impeller which main purpose is to translate the energy from the pump/rotor to the air being pumped by accelerating the fluid outwards the center of rotation.

2.3.1 Centrifugal impeller theory

Centrifugal impellers have been used in many different applications all the way back since early 1800's. The most famous application is the simple centrifugal pump. Other research reports that are dealing with development of ventilated brake disc design, are investigating the pumping performance with help from centrifugal impeller theory.

A centrifugal impeller is, like a ventilated brake disc, a construction that imparts kinetic energy into the treated fluid. This translated energy will affect the fluid to flow. The basic theory of this flow phenomenon works basically as it sounds; the centrifugal impeller (or the ventilation channels) slings the fluid out of the impeller by centrifugal force, see picture 2.X below from Evans (2014).

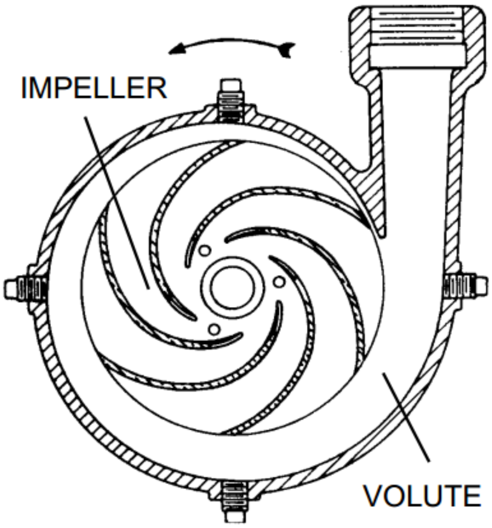


Figure 2.4: Drawing of a centrifugal pump.

The flow is pumped into the central circular inlet of the impeller and then push through the impeller flow channels. The flow is sucked into the inlet by the locally lower pressure.

2.4 Cool down behaviour

Newton's law of cooling is a useful theory when it comes to rating of different product cooling performances.

$$T(t) = T_2 + (T_1 - T_2)e^{-s \cdot t} \qquad \text{Eq. 2.14}$$

$$T_0 = T_{t \to 0} = T_2 + (T_1 - T_2)e^{-s \cdot 0} = T_2 + (T_1 - T_2) = T_1 \qquad \text{Eq. 2.15}$$

$$T_\infty = T_{t \to \infty} = T_2 + (T_1 - T_2)e^{-s \cdot \infty} = T_2 + 0 = T_2 \qquad \text{Eq. 2.16}$$

$$Eq. 2.2 + 2.3 \; into \; 2.1 \rightarrow \qquad T(t) = T_\infty + (T_0 - T_\infty)e^{-s \cdot t} \qquad \text{Eq. 2.17}$$

To calculate the s-value, also called cooling performance value, can later on be calculated by transforming equation 2.14.

$$\frac{T-T_\infty}{T_0-T_\infty} = e^{-s \cdot t} \quad \rightarrow \quad s = -\frac{ln\left(\frac{T-T_\infty}{T_0-T_\infty}\right)}{t} \quad , \qquad where \begin{cases} T_0 = 300 \text{ °C} \\ T_\infty = 25 \text{ °C} \end{cases} \qquad \text{Eq. 2.18}$$

This calculation can later on be used to get comparable quantities of cooling data of different brake discs and velocities.

2.5 Pressure

In aerodynamics and hydrodynamics, pressure is a commonly used physical property for explanation of flow behaviours. Three different pressure quantities will be presented in this sub-chapter.

2.5.1 Dynamic pressure

The dynamic pressure is often used as a quantity in flow contexts and can be described as the kinetic energy per unit volume of a fluid particle, see equation 2.19 below.

$$P_{dyn} = \frac{\rho V^2}{2} \qquad \text{Eq. 2.19}$$

2.5.2 Static pressure

Static pressure can be described by Bernoulli's equation which can be seen in Eq. 2.18 below. P_0 is the total pressure which is constant along a streamline which means that the sum of static and dynamic pressure is constant.

$$P_{dyn} + P_{stat} = P_0 \qquad \text{Eq. 2.20}$$

2.5.3 Pressure coefficient

In different flow situations, the pressure coefficient is often used as a dimensionless number to describe the flow behaviour which is a function that describes the relative free flow pressure for each point in the flow, see eq. 2.21.

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2} \qquad \text{Eq. 2.21}$$

2.6 CFD

Computational fluid dynamics or CFD as it usually is referred as, is a fluid mechanic division which uses algorithms and numerical methods to analyze and solve fluid flow problems. In order to perform all calculations needed for a given flow problem, computers are used. The flow problem can usually be described as the interaction between gases and liquids with surfaces defined by boundary conditions.

2.6.1 Turbulence model

"Turbulence is that state of fluid motion which is characterized by apparently random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena and results in increased energy dissipation, mixing, heat transfer, and drag." see CFD-Online (2014).

As good as all fluid dynamic engineering applications are turbulent and therefore require turbulence models to capture the correct behaviour. The most common turbulence models of today make a great key ingredient to many CFD simulations. There are three main classes of turbulence models; RANS-based models, large eddy simulations, detached eddy simulations (hybrid models) and direct numerical simulations.

One of the most common turbulent models is the K-epsilon model which is a two equation model in the RANS-based model class. This model has become very useful in many different industry applications even though it doesn't perform well in cases of large adverse pressure gradients. Since a rotating brake disc doesn't experience that high pressure gradients compared to a compressor-turbine case, this model will be tested later on for the CFD brake disc model.

Another commonly used turbulence model is the K-omega model. This is also a two equation model which belongs in the RANS-based model class.

2.6.2 Energy model

In previous chapter it has been known that the behaviour of the flow around a brake disc is temperature dependent, i.e. energy dependent. Normally when aerodynamic analyses are done and related forces are calculated, the energy isn't bothered in the simulation because of the low effect on the results. But in this case the energy is clearly necessary. The energy model therefore has to be applied in order to make Fluent solve the energy equation, see eq. 2.22 below taken from ANSYS (2014).

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{\text{eff}} \nabla T - \sum_j h_j \vec{J}_j + (\overline{\overline{\tau}}_{\text{eff}} \cdot \vec{v}) \right) + S_h \qquad \text{Eq. 2.22}$$

The energy equation above is described in a form that Fluent solves it in. The terms are described in the acronym chapter.

The CFD models used in this study will predict the heat transfer rate behaviour from the convection part of the total heat transfer due to the application of energy equation. This will exclude the fact that conduction and radiation might influence the flow and cool down behaviour inside and around the brake disc. All simulations will also be solved in steady state due to simulation effectiveness. If some external software like Radtherm would have been used parallel to Fluent, a transient load case would be more interesting due to the time changing behaviour of radiation and conduction.

2.6.3 Moving reference frame model

A moving reference frame model is very useful in many industrial CFD applications today. MRF models are steady-state approximations where individually specified cell zones in the domain are specified with a certain rotational and/or translational velocity. A local reference frame transformation is used between the pre-defined cell zones, see the interface in figure 2.5 below. This allows the flow variables in one zone to be used when calculating the variables at the interface of the adjacent zone.

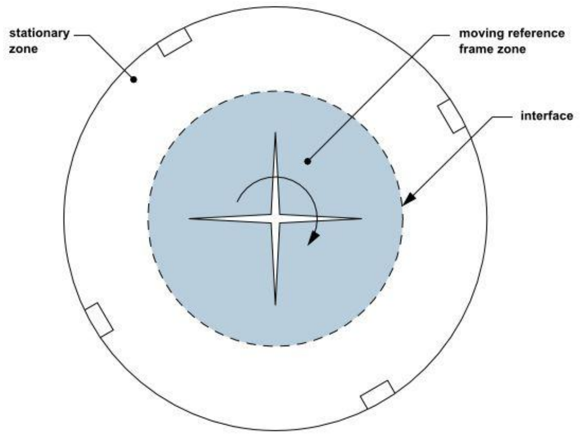


Figure 2.5: MRF rotating impeller example from ANSYS (2014)

It is also notable that the MRF approach doesn't take into account for the relative motion between one stationary cell zone and one moving cell zone (the moving MRF), which means that the velocity at the interface must be the same for both reference zones, the mesh interface therefore remains fixed during the simulation.

For a single rotating impeller mixing tank example, which is described in figure 2.5 above, also can be associated with a rotating brake disc case.

2.6.4 Flow definition

2.6.4.1 Density

For the flow definition, air is chosen as the fluid and cast iron is chosen as the solid brake disc material. The flow will depend on the density of the fluid and different ways of handle this can be defined in Fluent. The easiest and most effective way for Fluent of solving the energy equation is to set the density as constant. This behaviour doesn't really reflect reality due to the flow changes around the brake disc from a temperature and pressure point of view. Different kind of density approaches will therefore be tested in form of incompressible ideal gas law and ideal gas law, see equation 2.23, 2.24 and 2.25 below, see ANSYS (2014).

$$Constant\ gas{:}\ \ \rho_{const} = \frac{M_w \cdot P_{op}}{R \cdot T_{op}} \qquad \text{Eq. 2.23}$$

$$Incompressible\ ideal\ gas{:}\ \ \rho_{IIG} = \frac{M_w \cdot P_{op}}{R \cdot T} \qquad \text{Eq. 2.24}$$

$$Ideal\ gas{:}\ \ \rho_{IG} = \frac{M_w \cdot P}{R \cdot T} \qquad \text{Eq. 2.25}$$

The constant gas approach is therefore a simplified version of the IIG and IG approach where the ambient or the so called operating physical properties are used instead of the local relative temperature/pressure fields.

2.6.4.2 Velocity

To make sure the correlation between the brake test machine data and the imitative CFD model is valid, the measuring methodology has to be consistent. Fluent uses two different kinds of methods which are in interest of this study.

$$Facet\ average\ velocity{:}\ \ v_{avg} = \frac{\sum_{i=1}^{n} v_i}{n} \qquad \text{Eq. 2.26}$$

$$Area - weighted\ average\ velocity{:}\ \ v_{avg} = \frac{1}{A}\int v dA = \frac{1}{A}\sum_{i=1}^{n} v_i A_i \qquad \text{Eq. 2.27}$$

The facet average method which can be seen in equation 2.26 above computes the summation of the velocities over a surface of n number of facets divided by the number of facets. This is a simplified method compared to the area-weighted average methodology which divides the velocity summation of the selected field velocity and facet area by the total area of the surface. This methodology matches the method of the flow measuring device (anemometer: Schiltknecht MiniAir2) and will therefore be used for the post processing presentation.

2.6.4.3 Viscosity

When it comes to the viscosity definition of the flow in following models, the Sutherland viscosity law with three coefficients will be used, see equation 2.28 below from ANSYS (2014). For air at moderate pressures and temperatures; $S = 110.56\ K, T_0 = 273.11\ and\ \mu_0 = 1.716 \cdot 10^{-5}$.

$$\mu = \mu_0 \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S}{T + S} \qquad \text{Eq. 2.28}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

μ_0 = reference value in kg/m-s

T_0 = reference temperature in K

S = an effective temperature in K (Sutherland constant)

The viscosity law of Sutherland is based on the kinetic energy theory that Sutherland published in 1893 where he used an idealized intermolecular-force potential. This method has been used in many different thermodynamic applications at VCC with good correlation.

2.6.5 Initialization

Before any iteration can be introduced in Fluent for the CFD simulation to start, an initialization has to be made. It means that an initial "guess" has to be given for the solution flow field. Fluent uses three different kind of initialization schemes; standard initialization, FMG initialization and hybrid initialization. Hybrid initialization will be used in this study.

2.6.5.1 Hybrid initialization

Hybrid initialization uses boundary interpolation methods and a collection of recipes. Laplace's equation is first solved to provide a smooth pressure field between high and low pressure values and a velocity field for complex domain geometries. Other variables like volume fractions, species fractions, turbulence and temperature are automatically patched based on particular interpolation recipe or domain average values.

2.7 Analysis process

In order to simplify and speed up the whole CFD analysis process due to work overload, automated processes have replaced manual processes. At Volvo CFD/Aerodynamics department, a certain method of process is used in order to provide more simulations in a given time interval and chose each software strategically for each purpose.

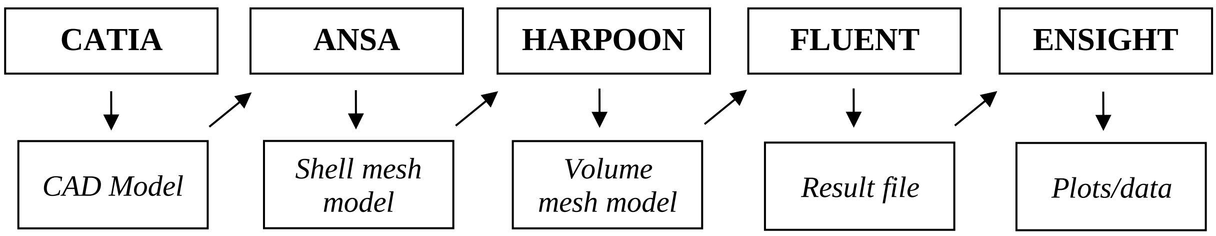


Figure 2.6: Schematic visualization of the simulation process

In figure 2.6 above, the whole process of simulation can be followed which is widely used at the CFD department of VCC. In Catia or optional CAD software, a detailed CAD model is created of the model and exported to Ansa. Dimensional correlations can also be done in Ansa but it is used mainly as a meshing tool for surface/shell meshing of the model. The work up to this point in the process is usually done manually. Thereafter a script can be used to automate the process. Next step is to write a script file which reads the shell mesh model, import it into Harpoon and performs a volume mesh according to the given mesh details. This volume mesh model is thereafter read by another script which exports the model to Fluent which is used as the simulating/calculating software. In this second script file the "load case" is defined and all the simulating properties are set. Depending on the comprehension of the simulation, the simulation takes between 10 minutes up to 10 hours. When the run is finished, the second script exports the result file which can be read and analysed by Fluent itself. This is normally not the case since Ensight is more preferable as post processing software. The result file is therefore exported into an Ensight result file which later on is evaluated in Ensight. Fluent also works fine as a post-processing tool for fast result confirmations.

Since a lot of model testing is necessary for this study, these processes will be run manually for a higher testing efficiency of new ideas and models. In a later stage, these processes can be automated by using CFD- Auto script, which takes care of everything from the importation of CAD model to the final exportation of post-processing data.

3 EXPERIMENTAL INVESTIGATION

In order to conclude that simplified simulations as CFD analysis can be considered as valid for the brake disc case, some kind of correlation with reality is needed. Due to the higher risk of being affected by external error factors when measuring an outdoor driving vehicle, other physical tests were instead used for correlation and more stable test data.

This chapter introduces the two chosen experimental equipment models to be used in this study, how the measuring was done and the result that came out of the different tests.

3.1 Volvo wind tunnel

Volvo wind tunnel is a state of the art wind tunnel which was upgraded in 2008 with the amount of 200 million SEK. The wind tunnel is driven by a huge fan with blades made of carbon fibre with a diameter of over 8 meters which can generate a wind velocity of up to 250 km/h. Volvo Car Corporation was the first car manufacturer in the world to build a own wind tunnel in 1986 and twenty years later the upgrade with rotating wheels, moving ground and larger fan, made the wind tunnel to one of the best wind tunnels in the world, see VolvoCars.com (2008).

With the quality of results that the wind tunnel provides, it can be concluded that the flow behaviour in the wind tunnel correlates well with the flow behaviour in a real life driving case.

3.1.1 Equipment description

The test was performed in the full scale wind tunnel of Volvo which is of a closed-circuit type with slotted wall sections by the test area (see figure 3.1).

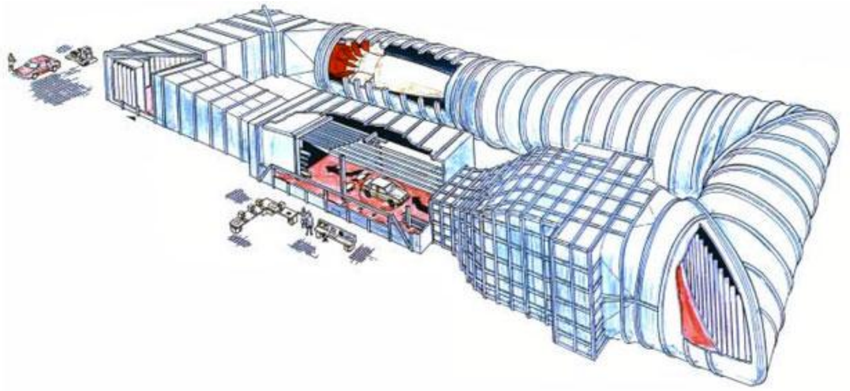


Figure 3.1: Sketch of Volvo wind tunnel, taken from STARCS (2014)

3.1.2 Test methodology

To get an idea of how the flow behaves in the wheel house of a full scale vehicle, Volvo wind tunnel was used to measure pressures in the wheel house of the mid-size Volvo model. Since this test data later on will be compared to the full vehicle CFD results of the same vehicle, prude investigations had to be considered in order to get a valid comparison of the data quantities and locations of the wind tunnel.

The main focus of this investigation is the air flow around and inside the brake disc, therefore a cutting plane through the cooling vanes would be valid as a comparison between the wind tunnel test and the CFD model. A cutting plane of the brake disc on the CFD model is no problem to achieve, but in the wind tunnel test, the pressure sensors must be placed in correct order to achieve a comparable plane. A laser measuring tool was used for this and the sensors could thereafter be placed in the right position of a reference plane through the brake disc, see figure 3.2 below.

Figure 3.2: Attached pressure sensors in in measured reference plane

Seven pressure sensors were attached inside the wheel house of the front right wheel with equal distances between. These sensors will later on represent a distributed pressure result of the wheel house inner surface. Later on the lower left sensor in figure 3.3, will be named sensor one and the upper next sensor two etc. Since the test is mainly a thermo-test for the engine area, the temperatures will be higher than usual between the steady state wheel house tests. The measuring tubes therefore had to be checked if they could manage the heat without jeopardizing the reliability of the results.

Figure 3.3: The seven pressure sensors inside the wheel house

3.1.3 Wind tunnel test data

The pressure coefficient was measured at three different positions inside the wheel house of the mid-sized Volvo model with start in the lower left corner, see figure 3.4 below. There after measuring point 2, 3, 4 etc. clockwise around the outer wheel house surface with approximately equal distances.

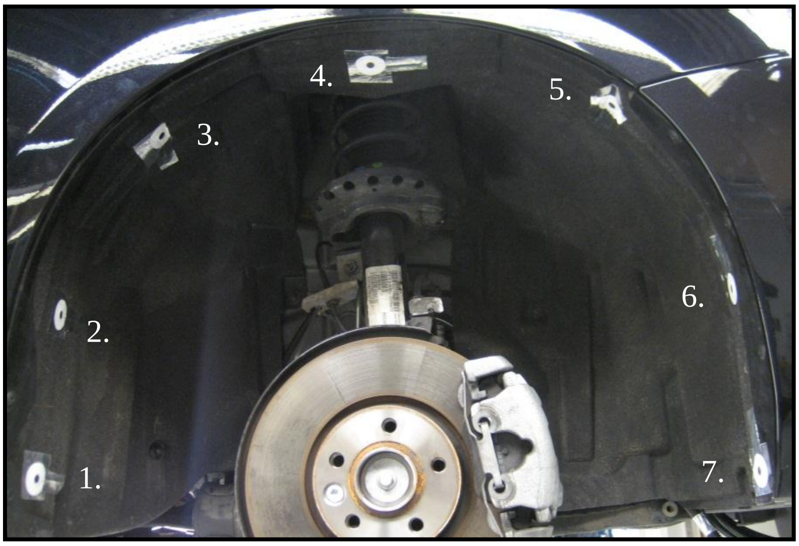


Figure 3.4: Measuring locations of the Volvo right front wheel house

The results from the wind tunnel test can be seen below in figure 3.5. It can be seen that the pressure coefficient values don't differ that much between the velocities except for the second measuring point where at higher velocities, the negative pressure coefficient decreases by approximately 17%.

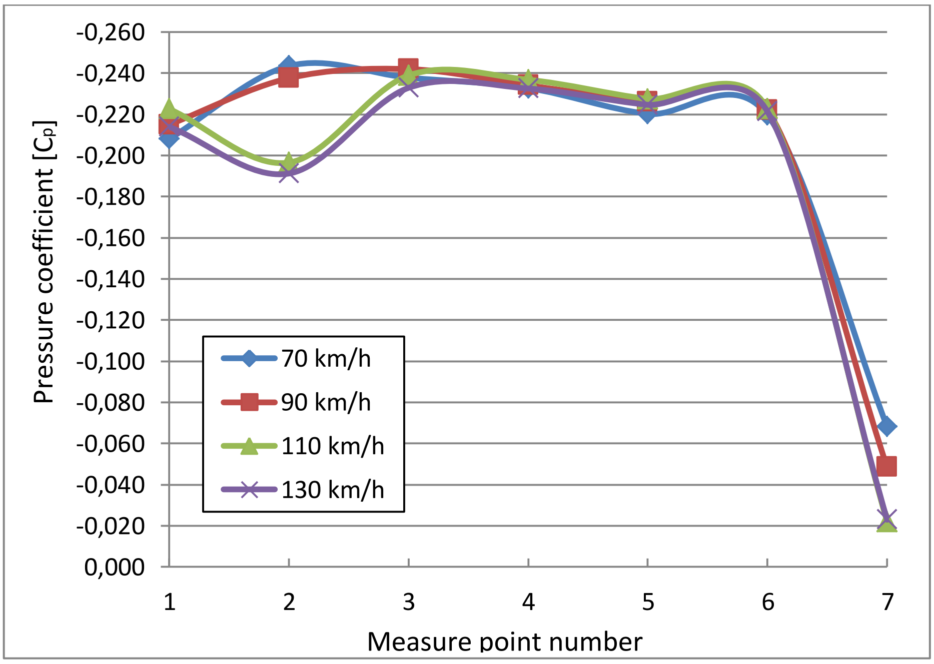


Figure 3.5: Wind tunnel test data – 4 different velocities – C_p at 7 different locations

3.2 Volvo brake machine

The brake test machine at Volvo Car Corporation headquarter in Torslanda, is a construction of a rotating counterweight which simulates the kinetic energy of a vehicle, the brake machine chuck which is attached to the counterweight as a fixation construction for the brake disc hub and the caliper construction on the opposite side.

As described previous in the report, the mass flow rate through the disc vanes (pump effect) and the cooling time are two important and connected properties. These will therefore be measured with flow measurement equipment, temperature sensors and caliper construction.

3.2.1 Equipment description

In order to get comparable measurements of the mass flow rate (pump effect) and the cooling rate of the brake disc, a caliper construction had to be made. The flow measurement equipment was attached without any caliper on the brake disc. In that way the pure pumping effect of the brake disc could be measured without including another parameter as the caliper. A caliper construction was therefore made in order to warm up the brake disc to the preferred temperatures, and then remove the caliper for the cool down/mass flow measurements.

The flow measurement equipment is a construction of a funnel (1), an anemometer (2) and an inlet nozzle (3), see figure 3.6 below. Closest to the rotor and the vane inlets is the funnel to make sure all measure mass flow is due to the pumping effect of the brake disc. Then the anemometer is attached inside of the funnel inlet with great fit. At the inlet of the anemometer, the inlet nozzle is attached to direct the inlet flow is order to create a natural pumping of the rotor and to avoid blockage of the flow in the nozzle.

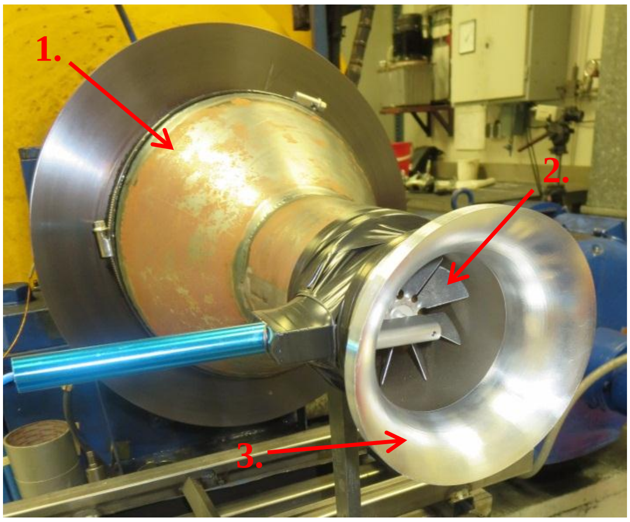


Figure 3.6: Volvo brake machine experimental equipment

The caliper construction was made out of aluminium and steel bars to get a solid, stable and moveable carrier of the caliper, see figure 3.7 below.

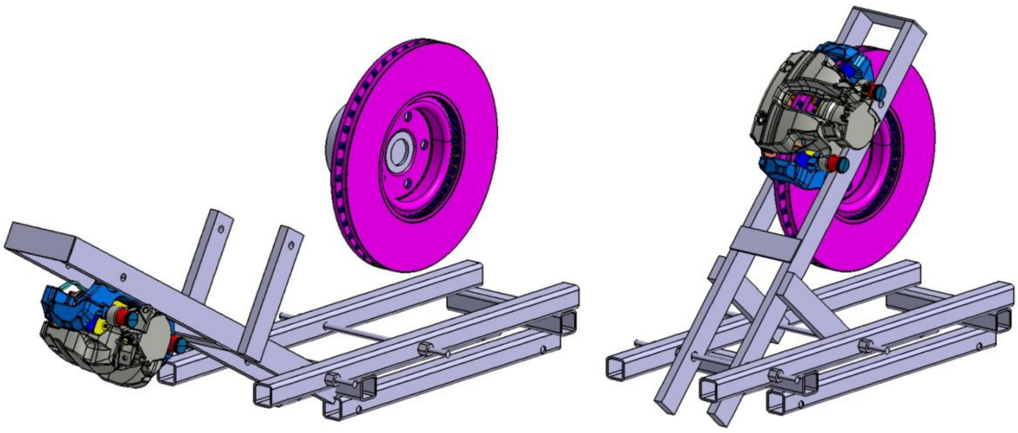


Figure 3.7: CAD model of the caliper arm construction – open and closed.

The brake disc is rotating clockwise seen from the view of figure 3.7, therefore the caliper arm construction was created in a way of loading the arm with tensile force instead of pressure force which has a higher risk of instability due to vibrations.

3.2.2 Test methodology

It's needed for the brake disc temperature to be as close to homogeneous as possible inside and around the brake disc to obtain comparable measurements when shifting rotor designs. Therefore a tolerance level of maximum 2 % is set for the temperature deviation of the inner friction surface and outer outlet vane wall.

When the disc temperature distribution criterion is reached and all temperatures exceed 300℃ during braking cycle, the caliper arm construction will be removed and the mass flow measurement equipment will be placed in position. Data for mass flow rate during cool down and cooling time from 300℃ to 100℃ will then be obtained for evaluation and comparison.

The two brake disc designs that are used in this experiment are two 18" front brake discs that look exactly the same besides the vane designs; pin design and straight ventilations channel design. In that way a fair comparison can be made out of a cooling vane design point of view.

Equipment:

- Specially manufactured air direction control nozzle for measurability of the pumping flow with included stance.
- Anemometer: Schiltknecht MiniAir2, signal generator: s/n 71253, borrowed from VCC-PVKA, recently calibrated.
- Specially manufactured inlet air nozzle for naturally inlet flow into the anemometer.
- Specially manufactured movable caliper arm for heating of the brake disc.
- Front right caliper, 18".
- Volvo brake test machine at Volvo Torslanda plant.

The tests were performed in the following steps:

1. Attach the caliper construction onto the brake disc and lock the arm by pushing the locking rod in its position and close it with the associated nut (see figure 3.8).
2. Slowly heat the brake disc by braking of the caliper in a constant velocity of 15 km/h and a caliper pressure of 1 MPa which represents a braking situation when stopping a car during parking.
3. When the heat distribution criterion is reached, the rotation is stopped, the caliper arm removed (see figure 3.9) and the flow measurement equipment attached in the correct position (see figure 3.10).
4. The measurement starts where temperature and pump flow data is loaded during the cool-down interval for a certain constant rotational velocity.

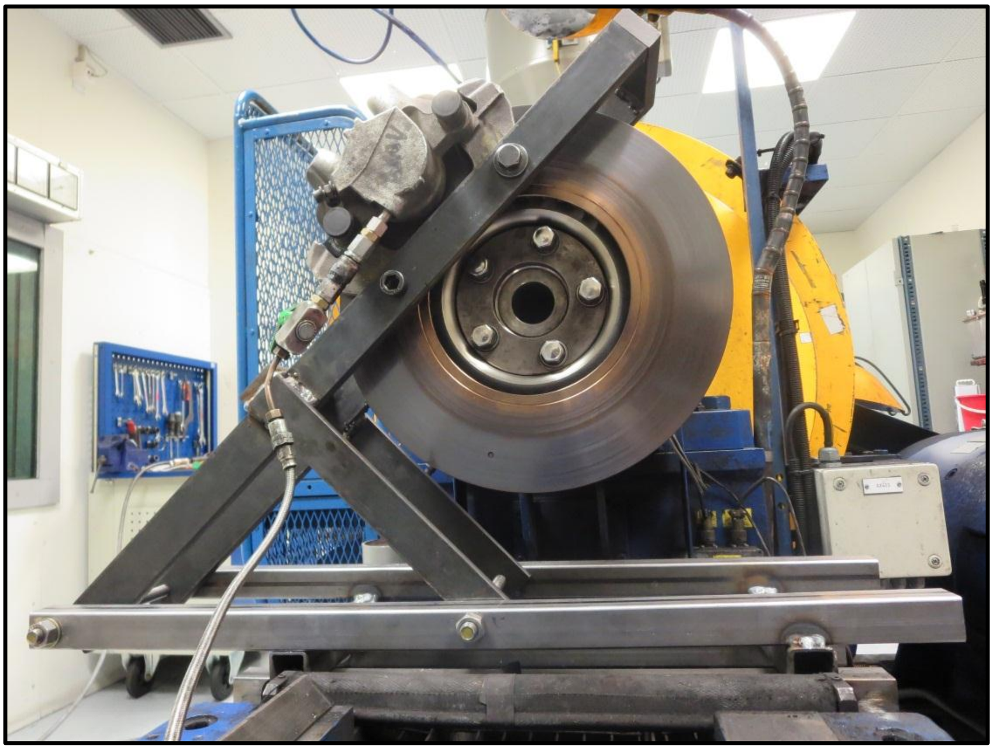


Figure 3.8: Locked caliper arm attached onto the brake disc

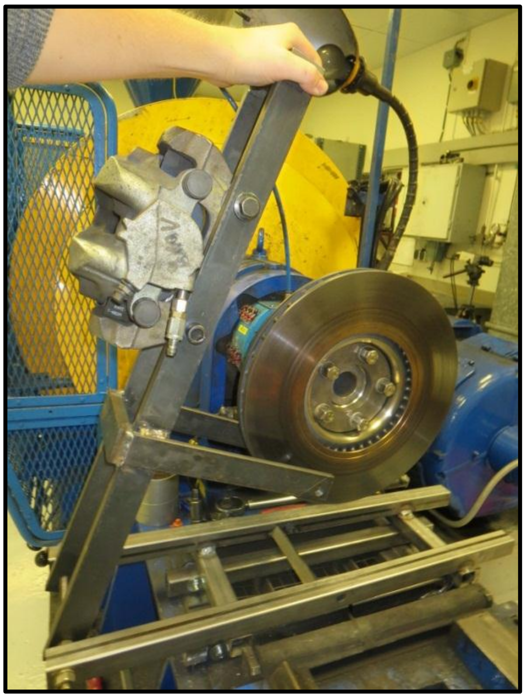


Figure 3.9: Removal of the unlocked caliper arm

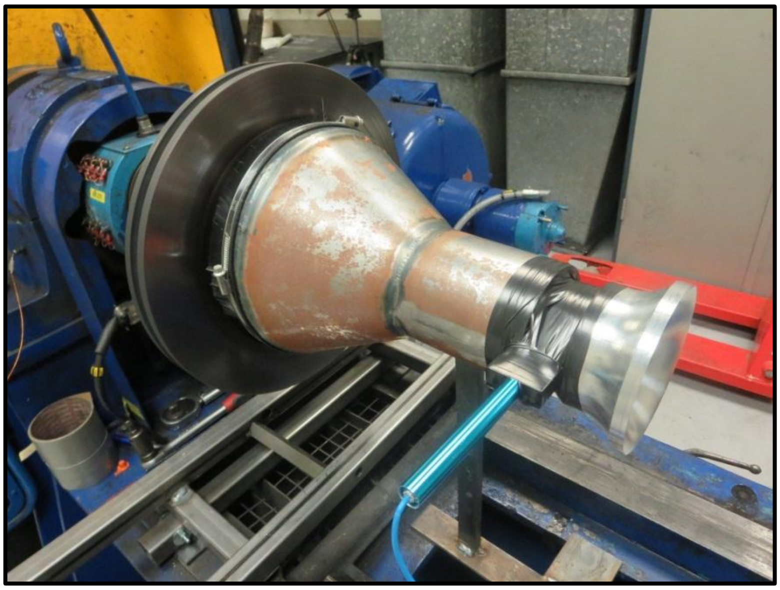


Figure 3.10: Attached flow measurement equipment

When the brake disc is heated to the preferred level the caliper arm has to be removed and the flow measuring equipment attached. For this procedure to be consistent, a pre-defined attachment position has to be set. Below in figure 3.11, a brush distance convergence study can be seen where it can be concluded that a distance of maximum 2 mm should be kept throughout the tests for consistent and comparable measuring data. Zero brush distance means that the brush is actually lagging onto the brake disc and thereafter the pumping flow is measured for each moved millimetre of the brush equipment. Both pin and vane design keep their flow deviation under 1.5% if the distance is kept under 2 mm.

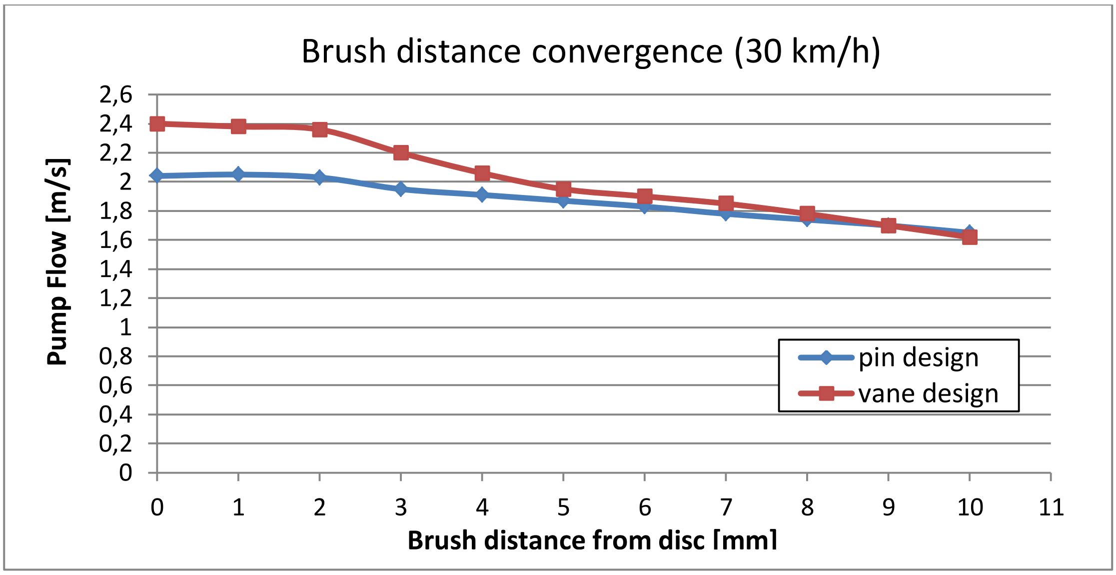


Figure 3.11: Brush distance convergence study at 30 km/h

3.2.3 Brake test theory

As can be seen in figure 3.X below, the cool down behaviours for 100 km/h and 150 km/h are approximately the same but for lower velocities the vane design has better cooling performances. It's notable that the two brake discs were weighted before the experiment and the result was 11.45 kg and 11.88 kg for pin and vane design, respectively. Therefore following conclusion can be made:

$$C_{p,pin} \cdot m_{disc,pin} \cdot \frac{dT}{dt}_{pin} = \dot{Q}_{in,pin} - \dot{Q}_{out,pin} = \dot{Q}_{net,pin} \qquad \text{Eq. 3.1}$$

$$C_{p,vane} \cdot m_{disc,vane} \cdot \frac{dT}{dt}_{vane} = \dot{Q}_{in,vane} - \dot{Q}_{out,vane} = \dot{Q}_{net,vane} \qquad \text{Eq. 3.2}$$

$$\begin{cases} Same\ material\ \rightarrow C_{p,pin} = \ C_{p,pin} = \ C_p \\ Both\ heated\ from\ 25\ ^\circ\mathrm{C}\ to\ 300^\circ\mathrm{C}\ \rightarrow dT_{pin} = dT_{vane} = dT \\ Mass\ difference\ \rightarrow m_{disc,pin} = \frac{11.45}{11.88} \cdot m_{disc,vane} \end{cases} \qquad \text{Eq. 3.3}$$

$$Eq. 3.3\ into\ Eq. 3.1, 3.2\ \rightarrow\ C_p \cdot \frac{11.45}{11.88} \cdot m_{disc,vane} \cdot \frac{dT}{dt} = \dot{Q}_{net,pin} \rightarrow$$

$$C_p \cdot \frac{11.45}{11.88} \cdot m_{disc,vane} \cdot dT = \ \dot{Q}_{net,pin} \cdot dt_{pin} = Q_{net,pin} \qquad \text{Eq. 3.4}$$

$$C_p \cdot m_{disc,vane} \cdot dT = \ \dot{Q}_{net,vane} \cdot dt_{vane} = Q_{net,vane} \qquad \text{Eq. 3.5}$$

$$Eq. 3.4 + 3.5\ \rightarrow\ Q_{net,vane} = 1{,}038\ \cdot\ Q_{net,pin} \qquad \text{Eq. 3.6}$$

Since the net heat is the absorbed heat energy of each brake disc, the derivation above shows (see Eq. 3.6) that the vane design rotor has 4 % more absorbed energy to get rid of to the surroundings compared to the pin design rotor.

3.2.4 Cool-down test data

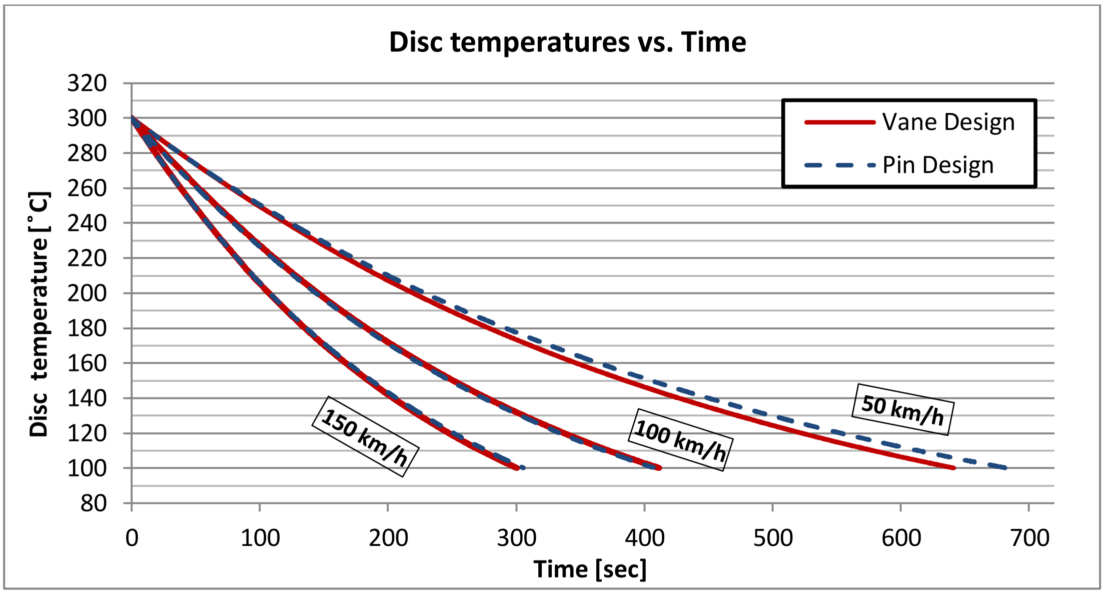


Figure 3.12: Cool down behaviour of pin and vane design for three different velocities.

A comparable cooling performance behaviour table can be obtained from the measurements above when applying equation 2.18 in previous chapter.

$$s(T,t) = -\frac{ln\left(\frac{T - T_\infty}{T_0 - T_\infty}\right)}{t}$$

Table 3.1: S-values for the different brake test machine configurations

S- values for brake test machine experiments ($* 10^{-3}$)			
Speed / **Disc**	50 km/h	100 km/h	150 km/h
Vane Design	2,03	9,05	12,10
Pin Design	1,90	9,04	12,10
Vane Design advantage	*6,84 %*	*0,11 %*	*0 %*

As can be seen below in figure 3.13, the theoretical behaviour explained by Newton's law of cooling agrees well with the experimental tests from the brake test machine.

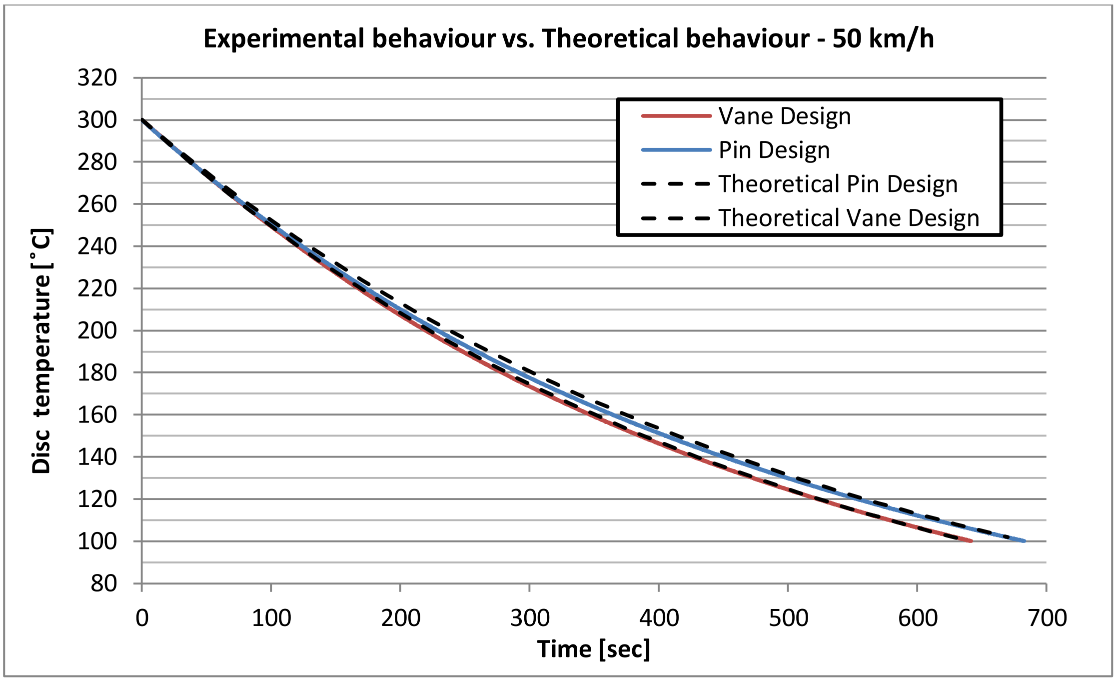


Figure 3.13: Visual comparison of experimental and theoretical behaviour.

3.2.5 Flow behaviour results

The outlet flow of the free rotating brake disc is complex and can behave in different ways depending on circumstances and dimensions. As can be seen in figure 3.14 below, the outlet flow behaves different depending on the measuring distance from the vane outlet. Close to the disc, the flow is directed in a circumferential direction as can be seen in lower figure 3.14. When moving the measuring thread outwards, the flow is directed in a more radial direction.

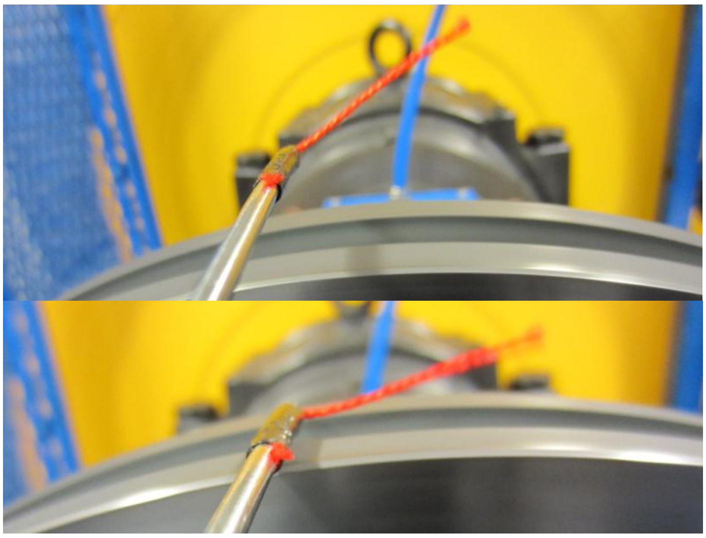


Figure 3.14: Outlet flow direction, two different measuring positions.

3.2.6 Flow test data

The temperatures from each time step is known from the cool-down tests in previous section and the pump-flow was measured for each time step, therefore pump flow vs. temperature diagrams can be plotted, see figure 3.15, 3.16 and 3.17 below.

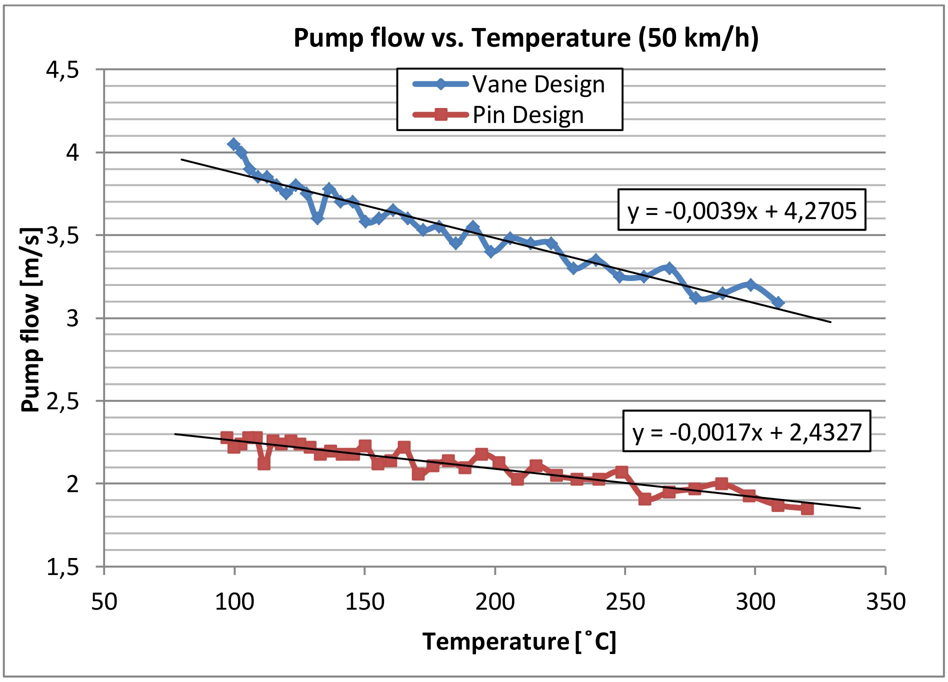


Figure 3.15: Pump flow vs. Temperature at 50 km/h

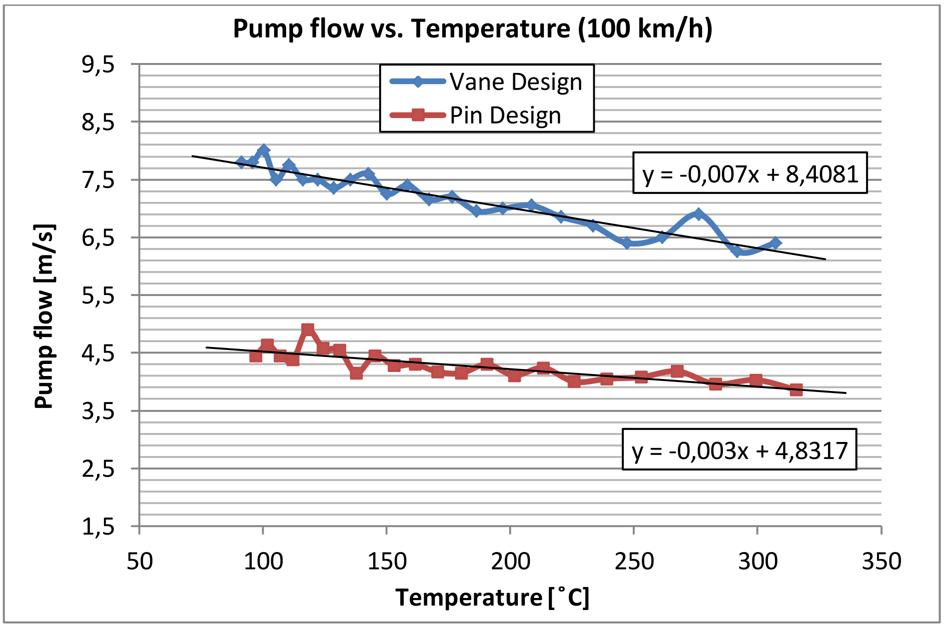


Figure 3.16: Pump flow vs. Temperature at 100 km/h

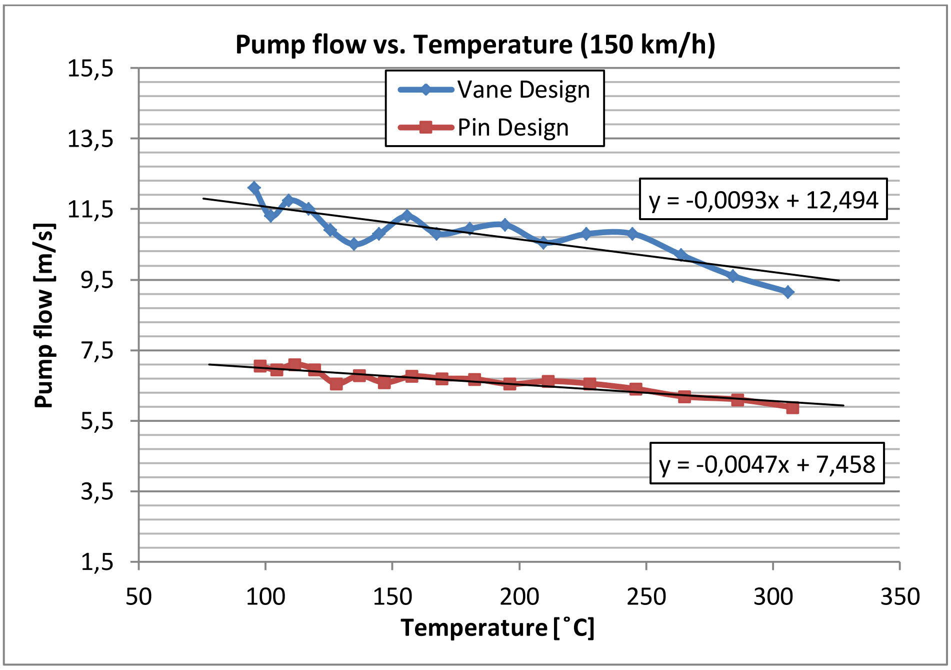


Figure 3.17: Pump flow vs. Temperature at 150 km/h

These diagrams represent the temperature dependent pumping flow for the chosen pin and vane design for three different free rotating velocities. During measurements, the flow varied and fluctuated at a certain level throughout the test. The measurements taken in the graphs above aren't therefore steady-state, but need to be linearized for a better correlation later on in this study.

4 NUMERICAL INVESTIGATION

Testing and verification of many industrial applications are today essential but usually expensive for the company. More and more company strategies go towards computer aided testing and verification instead in order to speed up the process and lower the costs. In many situations today CAE is preferred just because the test situation or load case cannot be verified in real life and therefore a numerical simulation is needed.

4.1 Full vehicle model

A real full-scale vehicle contains a lot of detail parts, which may or may not influence the primary result significantly. To be able to simulate a full vehicle case, simplifications have to be made in the modelling process for the simulation to run in a realistic time frame. The following chapter will present and explain these simplifications and boundary-conditions.

4.1.1 Modelling

This part of the study is made in purpose of having a correlating, trustworthy and well defined full vehicle model which can be compared both to a free rotating disc case and also earlier done experimental tests in Volvo wind tunnel. The vehicle used in this study is a mid-size Volvo model.

Table 4.1: Full vehicle parts with respective standard boundary-condition by VCC.

Part	Boundary-condition
Domain inlet	Velocity-inlet
Domain outlet	Zero pressure outlet
Domain ground	Moving wall
Domain sides and top	Symmetry (free slip condition)
Exterior and underbody	No slip wall
Wheel and rim	Rotating wall
Brake disc	Temperature and rotating wall
Space in rim spokes, brake disc vanes and cooling fans	MRF fluid zone
Space in radiator, charged air cooler and condenser	Fluid zones with different viscosity
Cooling package fans	Rotating wall

As explained earlier, Harpoon is used as the mesh generator from the exported shell mesh model from Ansa. A base level has to be set in Harpoon which describes what mesh size the standard level of each part will have. Then for each step-up of level, the surface mesh size will be halved which has to be defined for each part. The growth rate/mesh expansion, refinement boxes and size of the domain also have to be defined and set according to the requirements before Harpoon can generate a volume mesh. Volumes are defined by the surrounding surfaces of the volume which also defines the outer mesh size of the specified volume. In figure 4.1 below, the different volumes created by Harpoon in the full mid-size Volvo model domain are shown, besides the domain main fluid volume. Four different MRF volumes can be shown, one for each rim which will simulated the rotating flow behaviour inside the rime space. The volumes in the front represent radiator, charged air cooler, condenser and the cooling package fans because of their complex geometries. These volumes simulate the local pressure drops that occur and the rotating flow by MRF applications better than the domain flows would do. The internal brake disc MRF volume can also be seen in the right front wheel house in the figure.

Figure 4.1: Volumes defined in Harpoon of the full mid-size model.

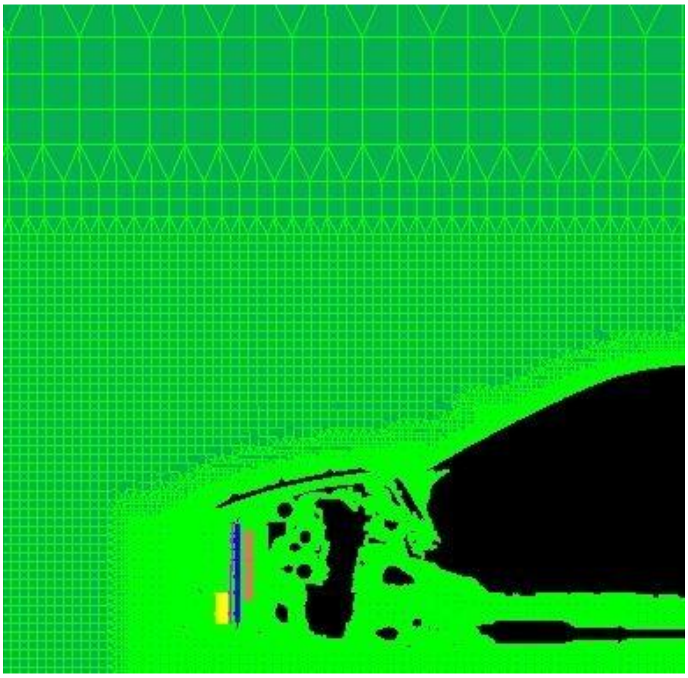


Figure 4.2: Domain volume defined in Harpoon of the full mid-size Volvo model.

In figure 4.2 above a cross sectional mesh plot is shown of the mid-size Volvo model. As can be seen, the mesh in refined close to the upper and lower chassis and the refinement boxes are refining the rear wake of the model, refinement boxes are also placed at the side mirrors. This model could also be run with closed front and would therefore look totally different at the internal front mesh, see figure 4.2 above.

In table 4.2 below, the different mesh sizes are shown. All relevant parts are defined with fine mesh definition of a hex-tetra mesh growth. The brake shield and caliper is defined with even smaller cells to make sure the caliper of brake shield grow together with the brake disc during the Harpoon volume mesh. If some parts would grow into each other and the actual space between the shield and disc disappears, the aero-thermal flow behavior around the disc would be affected and the model wouldn’t be reliable.

Table 4.2: Relevant wheel house part information

Part ID	Part name	Cell Size
1	Wall-brake-shield	0,5 mm
2	Wall-caliper-out	0,5 mm
3	Wall-caliper-in	0,5 mm
4	Wall-disc-mrf	1,0 mm
5	Wall-disc-rot	1,0 mm
6	Fan-mrf-inlet	1,0 mm
7	Fan-mrf-outlet	1,0 mm

4.1.2 Simulation data

The pressure coordinates were taken from the wind tunnel tests and then translated into the coordinate system of the CFD model to provide exact locations of the results in order to get a reliable comparison. As can be seen in table 4.3 below, all y-coordinates are the same since all points are in the same y- plane which has been measured during the wind tunnel preparation.

Table 4.3: Measure point coordinates

Point	Coordinate [x, y, z]
1.	2103, 784, 352
2.	2093, 784, 533
3.	1980, 784, 791
4.	1706, 784, 891
5.	1446, 784, 805
6.	1333, 784, 556
7.	1325, 784, 392

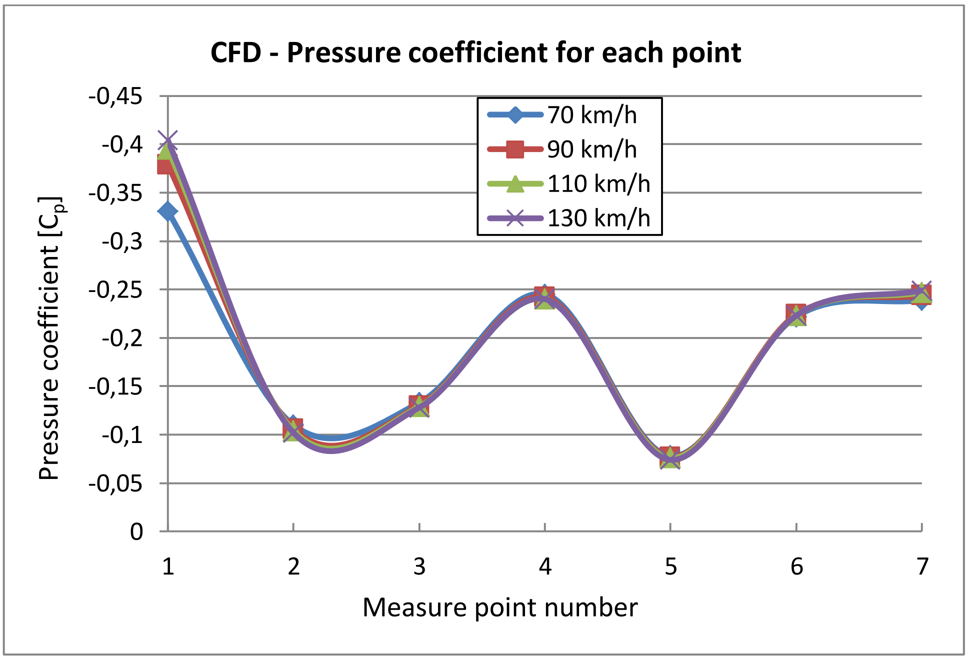


Figure 4.3: Pressure coefficients from CFD analyses at each measure point

The results from the CFD analysis can be seen in figure 4.3 above. It can directly be concluded that the deviation among the points are larger than the experimental data.

4.1.3 Residuals

After each simulation, the residuals and value convergences have to be checked, see figure 4.X below. It can be seen that the specific simulation in this case converge at around 4000 iterations. If the properties of interest look fine and have converged and the residuals are converged as below, the analysis is approved. 5000 iterations usually converge well for the full vehicle energy case.

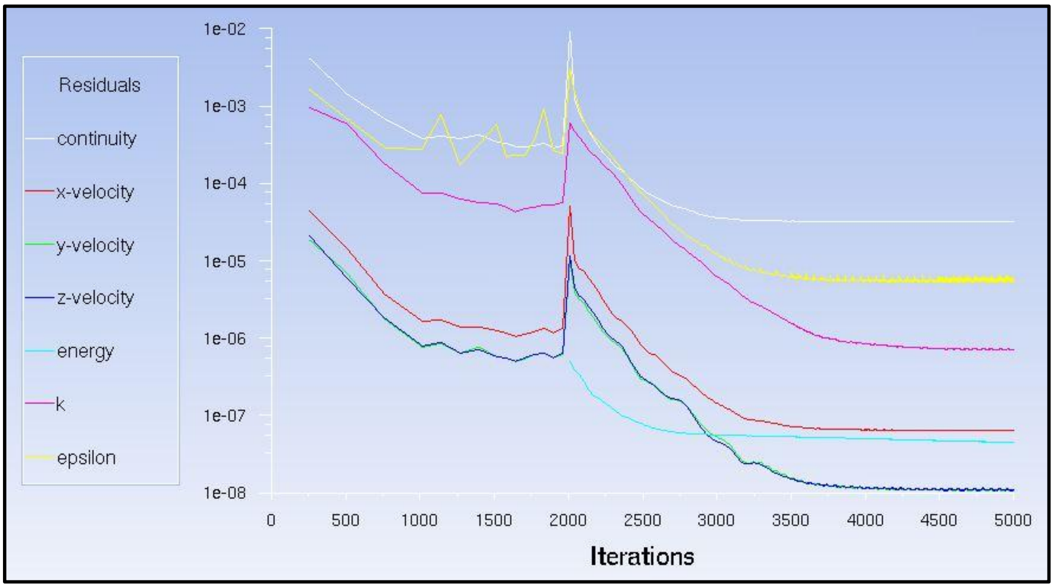


Figure 4.4: Example of all seven residuals

During the simulation/iterations, it can be pre-defined to export a certain flow property of interest for each iteration. This was made for each simulation, usually the mass-flow rate through the ventilation channels, to be sure that the properties of interest have converged even though the residuals aren't satisfactory. This means that the residual fluctuations depend on the convergence of a totally different part of the vehicle which is out of the scope of this study.

4.2 Brake machine model

The brake machine model is created in a way of mimic the experimental set up as good as possible in order to get a good correlation. In this chapter the whole process from measuring dimensions of the set up to analyzing the final simulation data will be explained.

4.2.1 Modelling

The model for the test equipment was constructed as a copy of the brake test rig except for some deviations, see figure 4.5 below. The measurement fan in the funnel inlet isn't modelled, just free space. The surroundings are also not modelled since the effect of the inlet and out pressures are assumed to be negligible. The funnel exterior isn't modelled exact since it has a very low effect of the result but the interior is of coarse modelled exact.

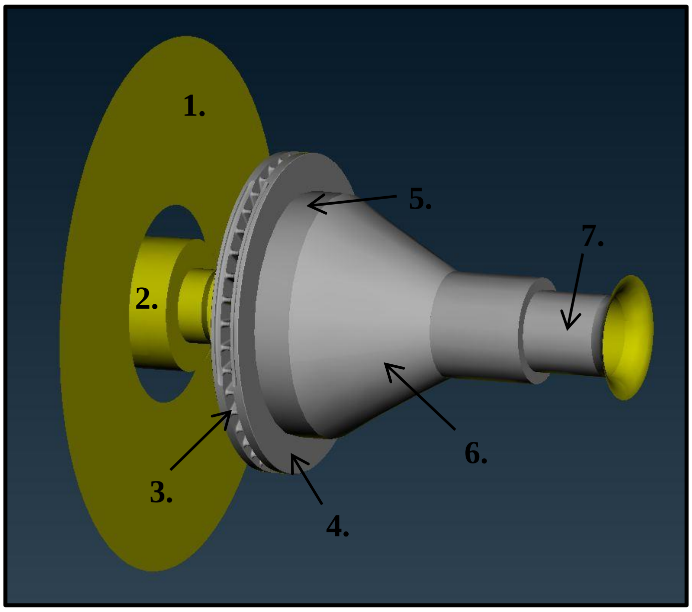


Figure 4.5: CAD model of the flow measuring equipment with part ID.

Table 4.4: Brake machine parts with respective boundary-condition.

Part	Boundary-condition
Domain inlet	Velocity-inlet
Domain outlet	Zero pressure outlet
Domain ground, sides and top	Symmetry (free slip condition)
Space in domain	Zero velocity fluid zone
Brake disc exterior (Part ID 4) and chuck (Part ID 2)	Rotating and no slip wall
Space in brake disc vanes (Part ID 3)	Rotating MRF fluid zone
Remaining equipment (Part ID 1, 5, 6 and 7)	Zero movement no slip wall

The MRF zone is defined by the internal surfaces of the brake disc, i.e. all the internal surfaces of the cooling channels which is inside the closed net volume, see below in figure 4.6. This MRF application is used due to the complexity of solving the flow problem inside the cooling vanes by using moving surface definition, as the external brake disc surfaces are defined by. There is also a mathematical definition of the relative movement between the defined surface and the adjacent fluid cells. Another method would be to define the whole brake disc surface as sliding mesh. Then the surface mesh is actually moving and demands a transient analysis. This type of simulation craves a lot more simulation activity and would be very ineffective if other alternatives are available.

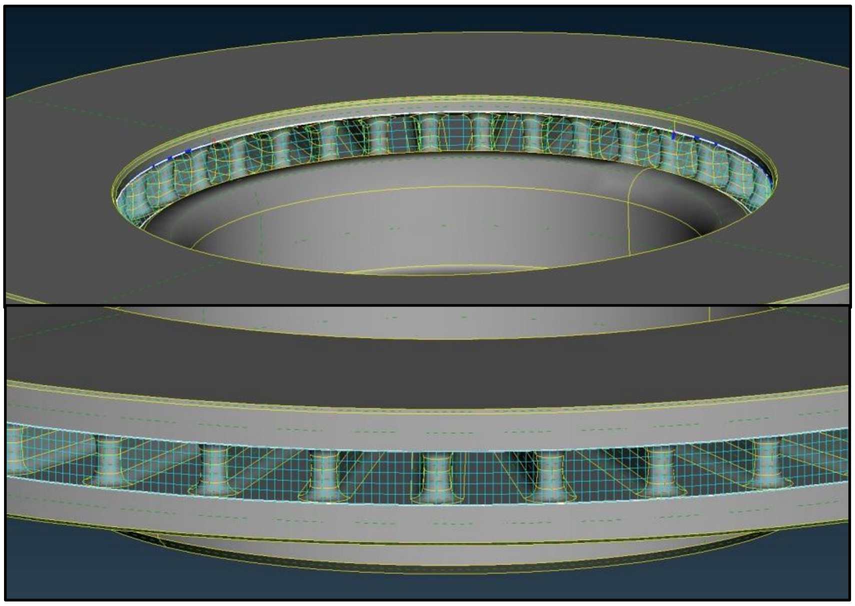


Figure 4.6: Rotating MRF- zone boundary

4.2.2 Meshing

The generated volume mesh which showed good result in the mesh convergence study can be seen as a middle cross section plot below in 4.7 and 4.8.

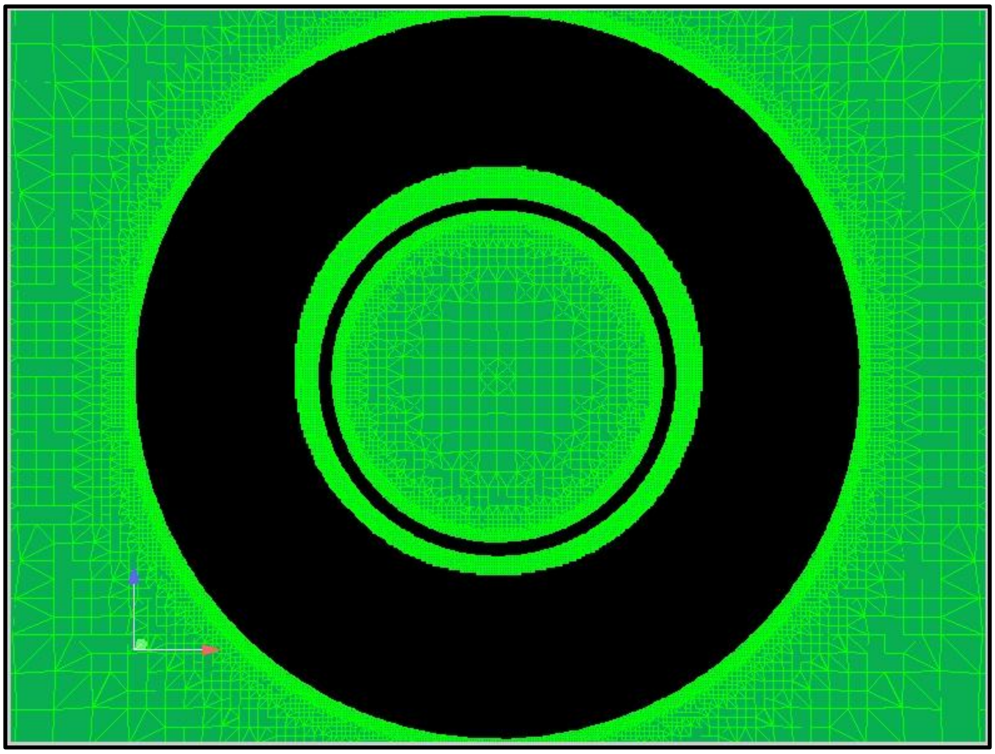


Figure 4.7: Brake disc cross section view of volume mesh.

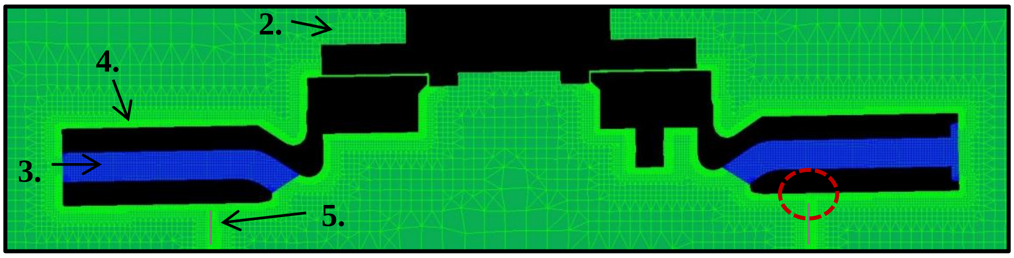


Figure 4.8: Brake disc cross section view of volume mesh.

To capture the flow behaviour from the experiment, the small distance between the brake disc and the measuring nozzle has to be valid, therefore and extra fine mesh is necessary in that region, see mesh size of the different part below in table 4.5 and marked area (dashed red circle) in figure 4.8. The surfaces that should be defined as solid walls are named wall and the other surfaces used for measurements and defining fluid volumes are named fan. See for an example in table 4.5 below, “fan-measure” is a fan- surface placed at the same position as the flow measuring anemometer was in the experiments and both “fan-mrf-inlet”/“fan-mrf-inlet” are two fan- surfaces defined at the inlet and outlet respectively of the brake disc ventilations channels.

Table 4.5: Information of parts.

Part ID	Part name	Cell Size
1	Wall-plate	2,0 mm
2	Wall-shaft-rot	2,0 mm
3	Wall-disc-mrf	1,0 mm
4	Wall-disc-rot	0,5 mm
5	Wall-nozzle-fine	0,5 mm
6	Wall-nozzle	1,0 mm
7	Fan-measure	1,0 mm
8	Fan-mrf-inlet	1,0 mm
9	Fan-mrf-outlet	1,0 mm

4.2.3 Limitations

This analysis will be set up in different ways according to the case running and the standard simulation procedures used at VCC. None of these simulations will be run transient or include any external thermal management software. The simulations will therefore not predict conductive or radiative thermal effects of the flow behaviour or the heat dissipating behaviour of the brake disc, only convective effects are treated by the energy equation.

4.2.4 Simulation data

The results for each velocity and temperature can be seen below in figure 4.9 and 4.10.

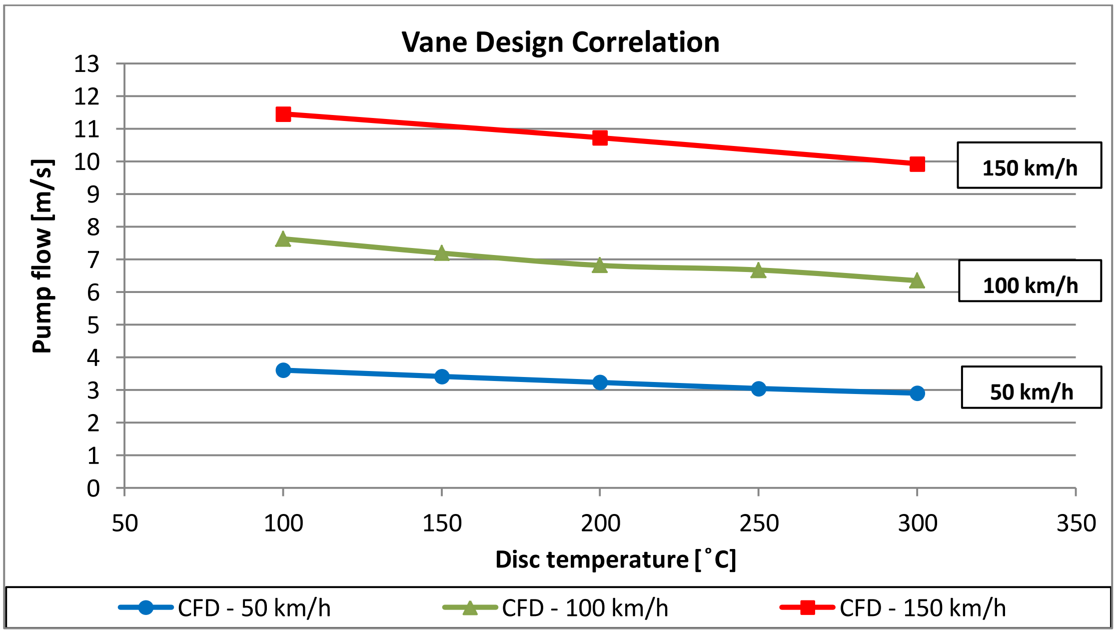


Figure 4.9: Simulation vane design data results for 50, 100 and 150 km/h

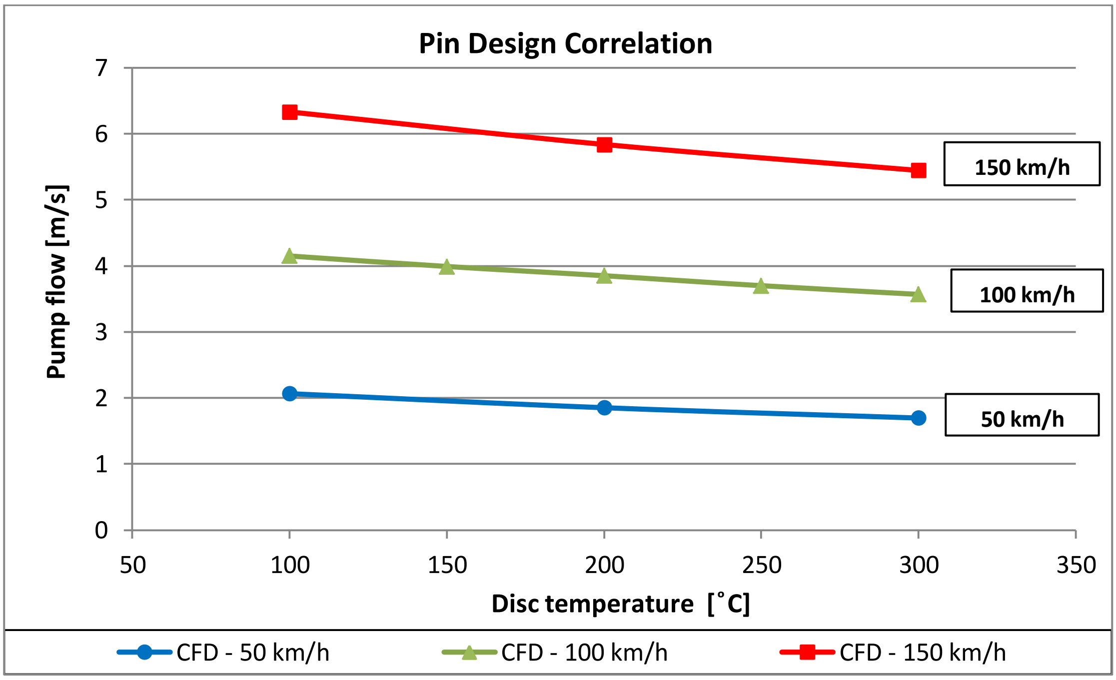


Figure 4.10: Simulation pin design data results for 50, 100 and 150 km/h

5 MODEL CORRELATION

To trust a model or a simulation, a correlation of some kind needs to be present in order to draw conclusions out of the results. This chapter will therefore handle the correlating/comparing part of this study.

5.1 CFD to Wind tunnel test

The full vehicle model has been developed by the Volvo aerodynamic team and has been correlating well with the full scale mid-size model test results. No physical testing was actually needed for this model since these full scale methods already has been evaluated. But some full model testing was interesting to perform in order to extend the overall knowledge of the area.

5.1.1 Pressure coefficient

From the earlier chapters where the data was obtained and evaluated, below is the assembled result diagram where a correlation evaluating can be made, see figure 5.1.

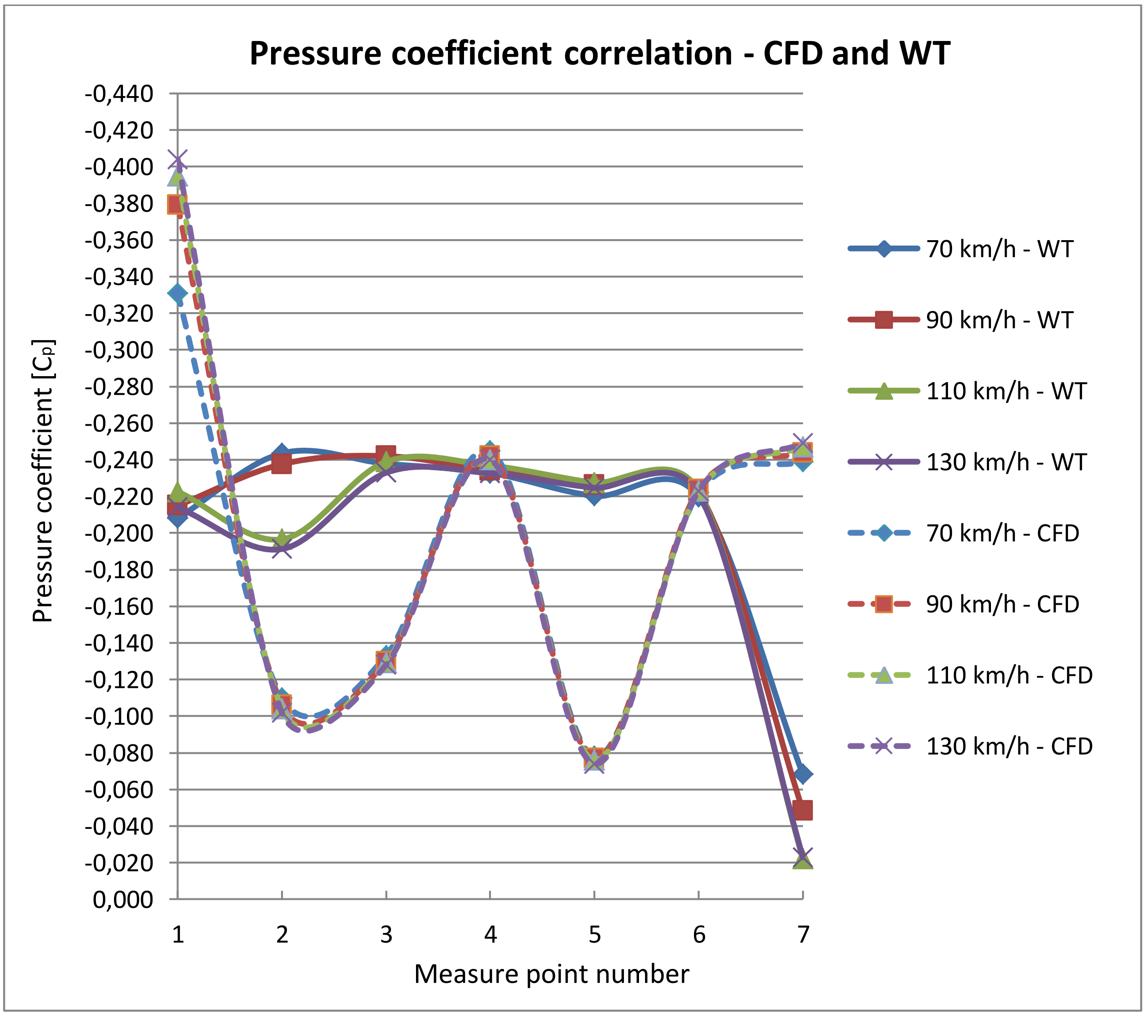


Figure 5.1: Comparison of pressure coefficient - wind tunnel experiment and CFD simulation

5.1.2 Conclusion

From figure 5.1, it can be concluded that there are definitely some deviations between the analysis data and the wind tunnel tests. Although, the main purpose of the correlation study for the full vehicle model was not to get an absolute decision of rating the usability, but rather of getting some hands on experience on the full vehicle flow behaviour in general. But some evaluation of the deviation appearance has to be made:

- The geometries don’t follow each other exactly due to the mesh size bigger than zero. Since the flow around the inner wheel house wall is very complex, the behaviour could be sensible to small geometry changes.
- The not exact locations of the pressure sensors could also provide deviations in the correlation since the analysis data was directly taken out of specific cells. These cells where placed on the surface of the centre where the representative location of the pressure sensor was attached, it could also mean that the sensor and fastener tape itself disrupted the actual pressure distribution along the surface.
- The analyses were simulated with defined moving ground boundary-condition to simulate the under-body flow behaviour which probably affects the wheel house flow behaviour a bit. The wind tunnel tests were performed with a moving ground in front of the car but not underneath since the car was placed on the dynamometer.
- Also a metallic safety net was placed outside the right front wheel in order to prevent accident in case of an emergency. The net probably affected the flow outside the wheelhouse which could affect the pressure distribution inside.

Otherwise, the full vehicle CFD model used in all analyses is overall used at Volvo and has been evaluated and approved. The only difference made in the model is the application of the brake disc flow model developed in this study to simulate the aero-thermal effect of the brake disc. So, the final conclusion is that even though the model didn’t correlate very well, the deviations can possibly be derived to any of the above mentioned suggestions, and will be continued as a reliable model because of its extensive use at VCC.

5.2 CFD to Brake machine test

5.2.1 Pump flow

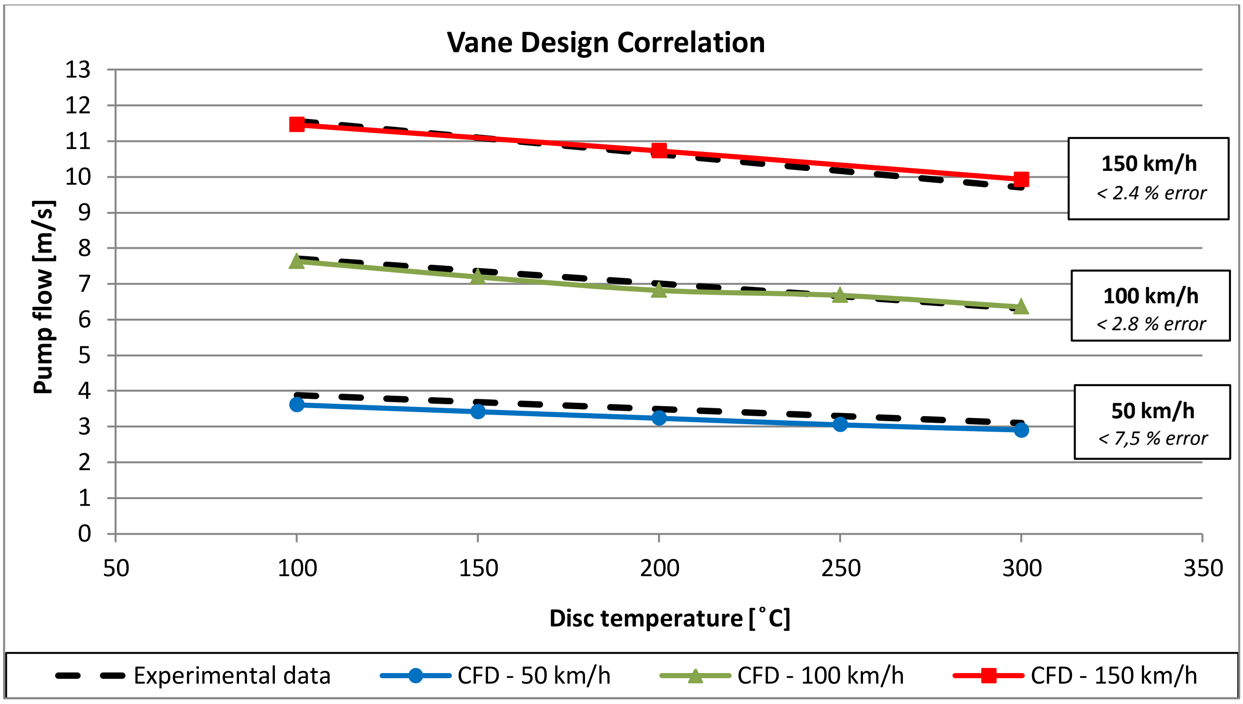


Figure 2.2: Correlation results of vane design model for 50, 100 and 150 km/h.

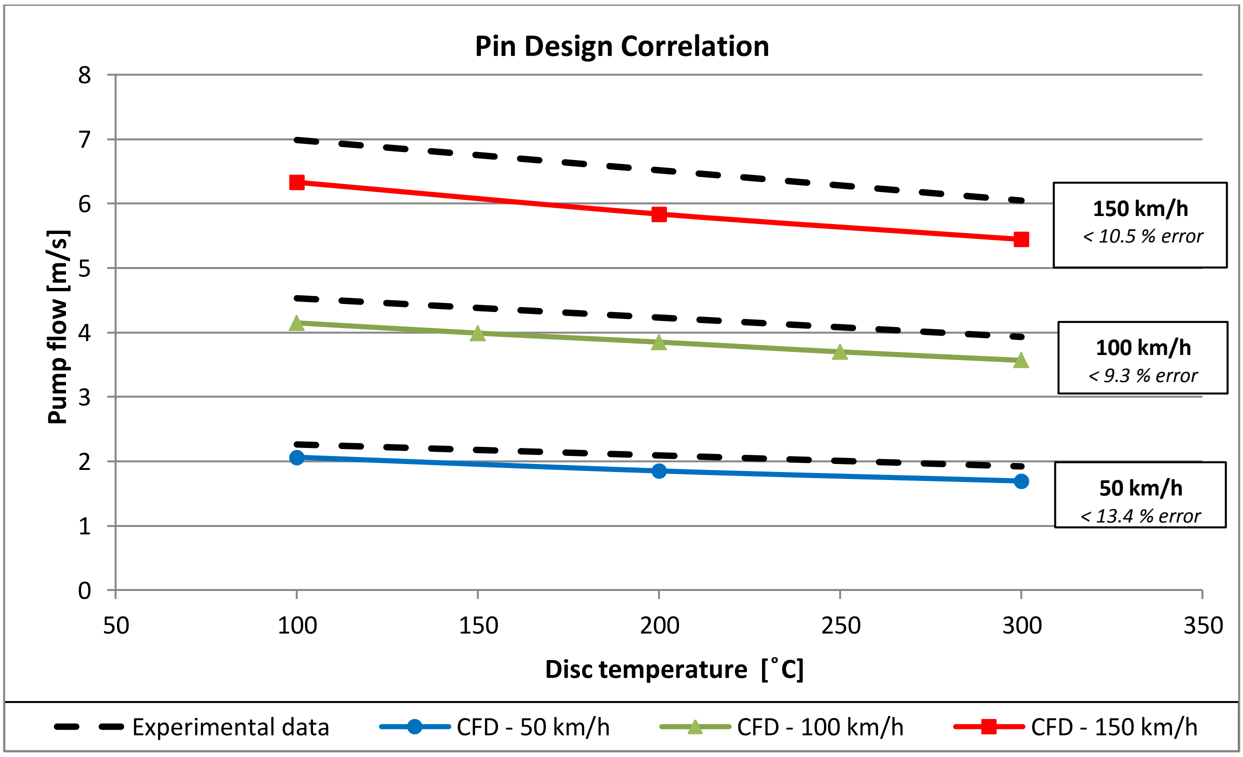


Figure 5.3: Correlation results of pin design model for 50, 100 and 150 km/h.

As can be seen in figure 5.2 and 5.3 above, the correlation results of the two models differ from each other. The pin design, which experience a higher turbulence rate because of the tougher channel flow geometry, does correlate with a deviation percentage of 9- 10% for higher velocities and even higher (13 %) for 50 km/h. It can therefore be concluded that the k-epsilon turbulence model used in this case doesn't predict the flow behaviour very well for the flow behaviour in pin design cooling channels, see figure 5.3.

When looking at vane design results in 5.2 instead, it can be seen that the model predicts the flow behaviour better. The max pump flow deviation for higher velocities is below 3%, and 7% for 50 km/h. This model therefore predicts the flow behaviour better than for the pin design. The vane design flow channel geometry is more pleasant from a turbulence point of view, which generally is a tougher property to predict in the world of fluid mechanics, the vane design is therefore easier to predict.

5.2.2 Visual flow

As an extension to the numerical data correlation between the CFD model and the brake machine experiment, a visual flow investigation was made in order to confirm that the vane outlet flow phenomena is behaving as it should.

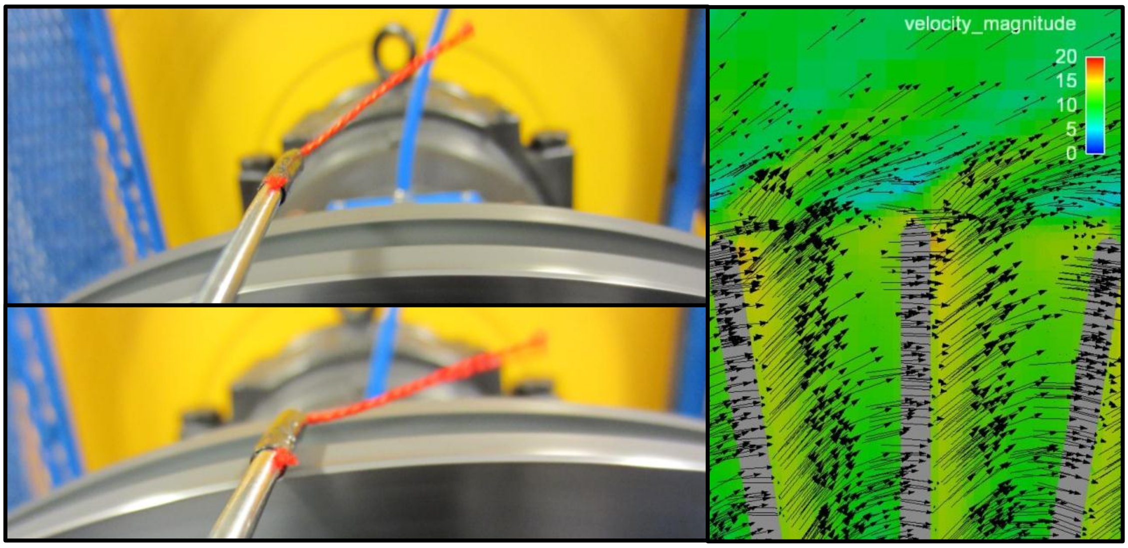


Figure 5.4: Experimental and numerical visualization of flow direction

As can be seen in figure 5.4, the measured/tested flow direction of the outlet vane flow correlates well with the CFD predictions. This zone by the vane outlets is the zone transition from the rotating MRF zone to the domain zone, which is extra important to check that it behaves as it should.

5.3 Conclusion

From what can be concluded out of the measurable properties and behaviour of the brake disc, the correlation was very good. To make the correlation more trustworthy, a higher level of experimental equipment is needed to ascertain the validity of the model. In the literature study, it was found that other engineers used e.g. small hot wire measurement equipment which provides the opportunity of correlating the flow velocities and pressures in the in- and outlets of the ventilation channels. Also a full thermal management model which simulates the radiation and conduction heat transfer to remaining parts of the experimental environment could be used. But comparing the comprehension of these methods to the already existing, this correlation is assumed to be enough for further investigation in use of this model.

6 CASE COMPARISON

In the automotive industry, an efficient and fast development process is always preferable due to the tough competition among car manufacturers today. This of course also applies to CAE verifications like these CFD simulations. This sub-chapter will therefore deal with the case similarities between the full vehicle model and the free rotating brake disc when it comes to flow and cool-down behaviour. The primary properties/behaviours that will be evaluated are the pumping flow at the measuring point, flow velocity at the cooling channel inlets, flow velocity at the cooling channel outlets, convective heat transfer from the different surfaces on the brake disc and flow direction inside and around the brake disc.

6.1 Flow behaviour

As can be seen in the flow plots in figure 6.1 below, the characteristic pumping flow behaviour can be seen both in the free rotating case and the full vehicle case. The view is at the front right wheel seen from the inside. It's important to clarify that this comparison is only valid for this specific case with a standard wheel house design of the mid-size Volvo model with a velocity of 100 km/h with a brake disc temperature of 200 degrees. As can be seen the plots aren't really symmetric since the planes are defined in the y- plane and the brake discs have a camber angle.

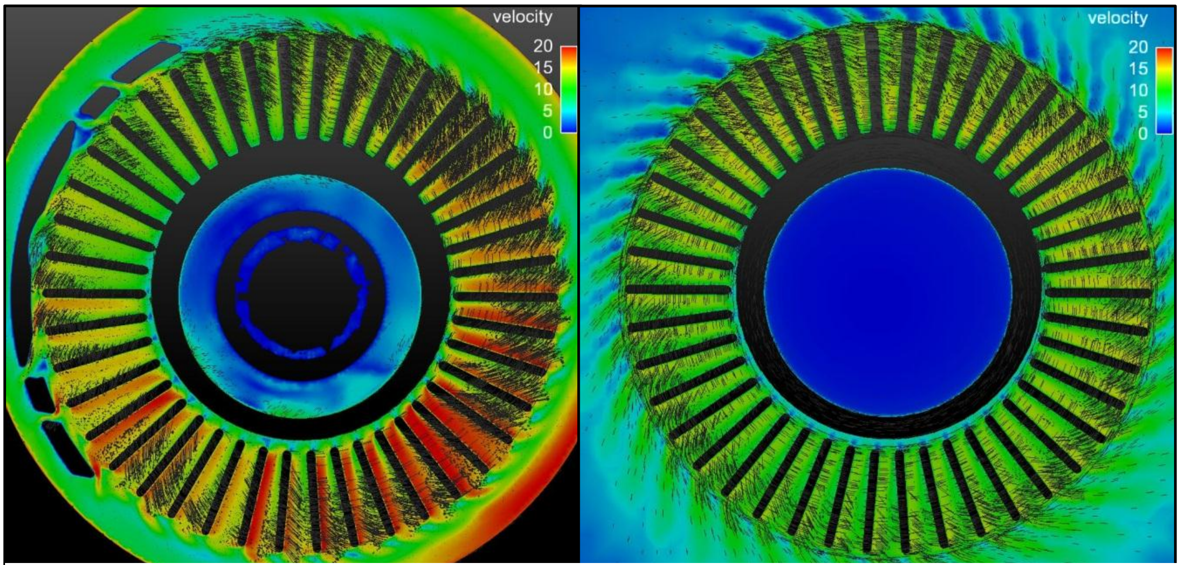


Figure 6.1: Visual flow velocity comparison of vehicle case to the left and free rotation case to the right.

It can be seen that the pump flow behaviour in the vehicle case to the left above is much more insymmetric compared to the free rotating case. The caliper is disrupting the pumping flow while the external inlet-flow is contributing to an increased flow rate at the "free edge" of the brake disc, see lower right corner in the left picture. The pump flow of the full vehicle case is 52.8 g/s and the free rotating case is 41.1 g/s. Even though this insymmetric flow behaviour appears in the full vehicle case, a high free rotating pump flow performance is still comparative applicable in the full vehicle case, see upcoming chapter 6.2.

6.2 Design correlation

Increasing the friction disc thickness, and also decreasing the vane gap, is a common brake disc design modification. When a specific brake disc design doesn't pass the crack tests requirements or the stiffness requirements, this modification comes in use. A big disadvantage of redesigns like this is the increased weight which is an important aspect in automotive applications. This redesign procedure is here illustrated as an example of the performance differences of the two brake disc in order to compare the relative performances both in the free rotating case and in the full nominal vehicle model with no wheel house modifications, see table 6.1 below. The cases used in this investigation are during a pace of 100 km/h and a disc temperature of 200 degrees.

Table 6.1: Disc design table

Disc design	Design description
1.	Nominal vent-in disc design, vane gap of 11 mm
2.	Modified vent-in disc design, vane gap of 8 mm

6.2.1 Comparison results

The different results for the modifications, cases and also the deterioration values can be seen in table 6.2 below.

Table 6.2: Disc design result table

Case	Vane heat flux [W]	Mass flow [g/s]
Disc design 1. – Free rotating	1316.2	41.1
Disc design 2. – Free rotating	1076.8	26.4
Deterioration	*-18.2%*	*-35.8%*
Disc design 1. – Full vehicle	1792.3	52.8
Disc design 2. – Full vehicle	1544.4	40.0
Deterioration	*-13.8%*	*-24.2%*

6.2.2 Conclusion

When looking at the results in table 6.2 it can be seen that the assumption of a representative simplified model was justified for this case. Even though the deteriorations aren't exact, they're in the same range and direction. The conclusion is that if the simplified free rotating case is used as a vane design developing tool, it should be considered that it overestimates the deterioration between the investigated designs but the application is valid. The simplification is applicable because the pump effect is detected in the full vehicle case, e.g. no reversed flow or major disruptions are found in the vanes, even though it's not as symmetric as the free rotating case. This conclusion is also limited to the specific case used in the investigation; speed of 100 km/h, brake disc temperature of 200 degrees.

7 DESIGN ANALYSIS

This chapter will handle an investigation where possible design alternatives are discussed that influences the cooling performance properties which earlier were concluded in this report.

7.1 Competitor investigation

When looking at competitor wheel house designs, a couple of things can be concluded.

Most of the competitors have no “deliberate openings” in the brake shield or the wheel house, they exist but they are few. Renault uses flow ducts right in front of the front wheel houses in the Renault Sandero model to direct the cool flow towards the brake disc. When it comes to front wheel deflectors, many cars today use some kind of wheel deflector. When looking at the popular car models BMW 328i, 523i and 740i, it can be seen that they are using front wheel deflectors but compared to the mid-size Volvo deflector, more aerodynamic design choices have been considered.

The general view is that most brake shields in the market today show approximately the same design strategy as the mid-size Volvo model does. A brake shield that covers the shield from dirt but at the same time letting some air inside towards the brake disc, some car models use a more covering shield design and some a more open design.

7.2 Brake shield design investigation

The brake shield is a wheel house part that is closely placed on the inside of the brake disc, see figure 7.1 below. The main purpose of the brake disc is to cover the brake disc from dirt which would interfere with the brake pad-brake disc contact patch and therefore also interfere with the brake performance of the vehicle. Seen from a cooling point of view, the brake shield covers the brake shield from convective heat transfer from the possible air flow, which is preferred in order to avoid an overheated brake disc state. The design of the brake shield could therefore be considered as a trade-off between cooling performance and dirt protection.

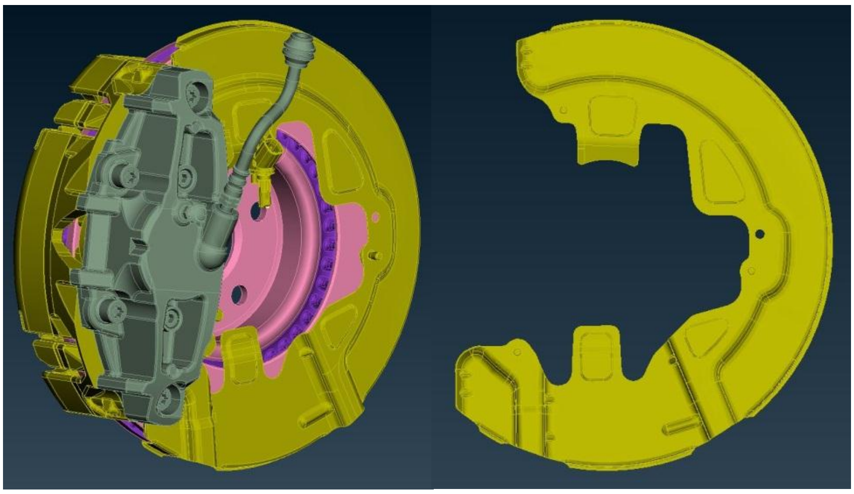


Figure 7.1: Nominal design of Volvo brake shield.

The brake shield is included in this study because it's the easiest part to modify to influence the cooling performance of the brake shield without interfering with other automotive areas.

7.2.1 Design modifications

The first design modifications tested were the two absolute cases, totally removed brake shield and totally covered brake shield. Even though these modifications aren't directly possible considering interference with other functions, this will provide a clear view of how much the brake shield actually influences the cooling performance of the brake disc.

A couple of other alternatives will also be presented and evaluated, see table 7.1 below.

Table 7.1: Description table of the different modifications

Appendix /Modification no.	Modification description
A0	Nominal model
A1	Removed lower front corner of brake shield.
A2	Removed lower half of brake disc and decreased upper part width.
A3	Removed outer lip and displaced brake shield +5 mm apart from brake disc.
A4	Closed outer rim.
A5	Totally covered brake shield and brake shield outer lip.
A6	Removed front wheel deflector.
A7	Removed brake shield.
A8	Removed front wheel deflector and brake shield.
A9	Open flow directory channel and removed brake shield
A10	Vent-out brake disc design, nominal wheel house design

7.2.2 Limitations

It should be noted that all mentioned modifications aren't actually physically possible with the present wheel hose/underbody/suspension design. A few designs are made just to get a design overview of the change in the aero-thermal behaviour around the brake disc. For example, the fastener points for the brake shield aren't considered for designs A2 and A3, other sensors that use the brake shield as attachment cannot function or be placed with the brake shield removed and the totally covered brake shield might interfere with other suspension articles.

7.2.3 Results

The results from the design modifications and the nominal design of the mid-sized Volvo are shown below in table 7.2 and figure 7.2. A0 is the nominal model and the percentage deviation from that is calculated for all the modified cases A1 to A10.

Table 7.2: Results from the different modification simulations at 100 km/h

Appendix /Modification no.	Modification description	Convective heat flux [W]	Convective outer heat flux [W]	Convective vane heat flux [W]	Pump flow [g/s]
A0	Nominal model	**3289.4**	**1497.1**	**1792.3**	**52.8**
A1	Removed lower front corner of brake shield.	**3398.2** (+3.3%)	**1594,4** (+6.5%)	**1803.8** (+0.6%)	**53.4** (+1.1%)
A2	Removed lower half of brake disc and decreased upper part width.	**3551.4** (+8.0%)	**1708.5** (+14.1%)	**1842.9** (+2.7%)	**52.8** (+0%)
A3	Removed outer lip and displaced brake shield +5 mm apart from brake disc.	**3339.1** (+1.5%)	**1604.8** (+7.2)	**1734.3** (-3.2%)	**49.1** (-7.0%)
A4	Closed outer rim.	**2252.1** (-31.5%)	**866.5** (-42.1%)	**1385.6** (-22.7%)	**39.2** (-25.8)
A5	Totally covered brake shield.	**2448.9** (-25.6%)	**1378.5** (-7.9%)	**1070.3** (-40.3%)	**23.1** (-56.3%)
A6	Removed front wheel deflector.	**3191.6** (-3.0%)	**1554.3** (+3.8%)	**1637.2** (-8.7%)	**44.9** (-15.0%)
A7	Removed brake shield.	**3561.1** (+8.3%)	**1735,7** (+15.9%)	**1825.4** (+1.8%)	**51.7** (-2.1%)
A8	Removed front wheel deflector and brake shield.	**3654.1** (+11.1%)	**1930,9** (+29.0%)	**1723.2** (-3.9%)	**44.8** (-15.2%)
A9	Open flow directory channel and removed brake shield	**3695.3** (+12.3%)	**1953.4** (+30.5%)	**1741.9** (-2.8%)	**46.1** (-12.7%)
A10	Vent-out brake disc design, nominal wheel house design	**2613.3** (-20.6%)	**1365.7** (-8.8%)	**1247.7** (-30.4%)	**14.4** (-72.7%)

7.2.4 Design conclusion

The first thing to conclude about the different modifications that can be seen in table 7.2 above is that the rim openings are of big importance for the brake disc cooling. When looking at modification A4 it can be seen that the total convective heat flux decreases by 31.5% when the rim is closed. Another interesting comparison is the extreme cases of brake shield design strategy; totally closed and removed. Modification A5 embodies the results of the totally covered brake shield which shows a 25.6% decrease in total heat flux. Comparing this to the opposite case, removed brake shield, it can be seen that the other extreme case doesn't give the opposite results. The increase in total heat flux becomes only 8.3% in modification A7 due to the disability of brake disc pumping, see vane heat flux of only 1.8% increase.

It can also be concluded that all modifications where a major "open up" has been made, e.g. A6, A7, A8 and A9, i.e. increasing the mass flow towards the brake disc, the vane cooling performance doesn't benefit very much due to the increased intensity of the surrounding flow. The centrifugal impeller theory correlates to this phenomenon since the low pressure vane inlet doesn't pump the intense turbulence flow as good as the less intense flow. In all cases except A5, A8 and A9, the vanes are contributing to the highest convective heat flux part, which means their ability to function correctly should be as much prioritized as the outer total heat flux, see figure 7.2 below.

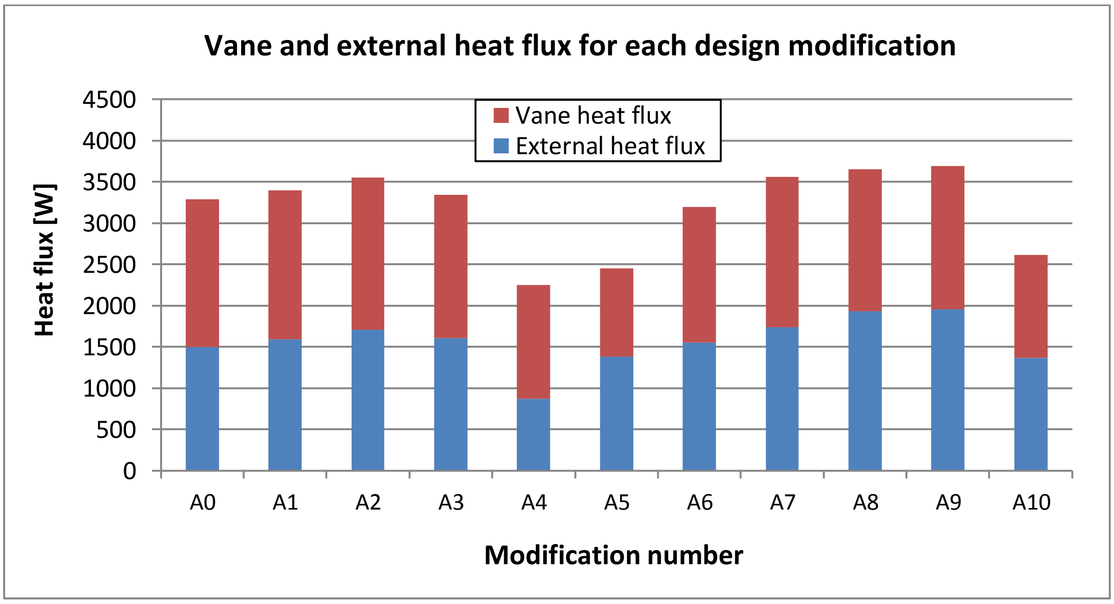


Figure 7.2: Bar schedule visualization heat fluxes of the different modifications

This means that a cooling performance design improvement doesn't necessary mean to increase and open up the flow towards the brake disc, but direct the flow in a strategic way that benefits both the external and internal heat flux of the brake disc.

The last modification, A10, represents the exchange of brake disc design from vent-in to vent-out. As can be seen, the total convective heat flux decreases drastically because of the really low vane flow activity (more than 30% decrease in heat flux and a pump flow decreases more than 70 %). This is because the general flow structure through the wheel house of this mid-sized Volvo is from the inside/underneath the car, through the brake disc, caliper and rims, and out. This through flow is also very important for the cooling effect of the brake disc which can be seen at A5 where the through flow pattern is blocked and the heat flux decreases drastically. For the vent-out brake disc design, this becomes a problem when the vane inlets are directed outwards, the flow path has to take a detour around the shield and then in through the vane inlets, to then also interfere with the vane outlet flow, see figure 7.3 below. It can also be seen that almost all inlet pump flow comes from the inside of the car, around the brake shield, and not from outside through the rims.

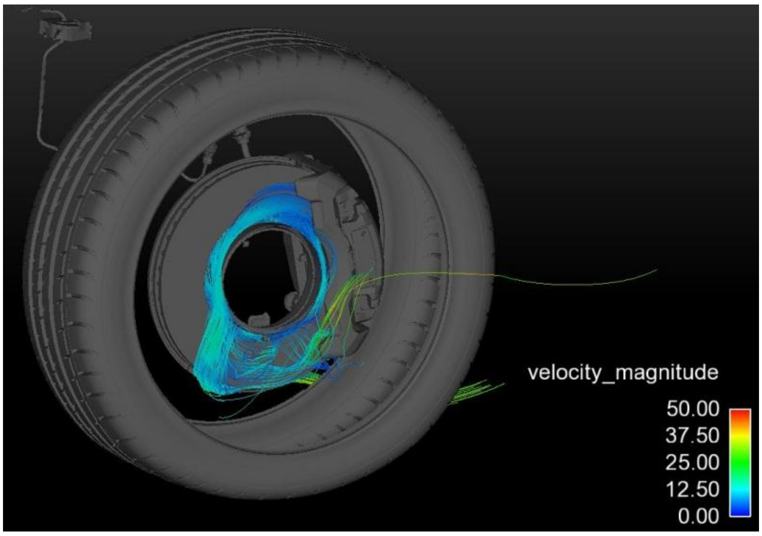


Figure 7.3: Inlet flow visualized, all parts masked except brake disc and shield, caliper construction and wheel.

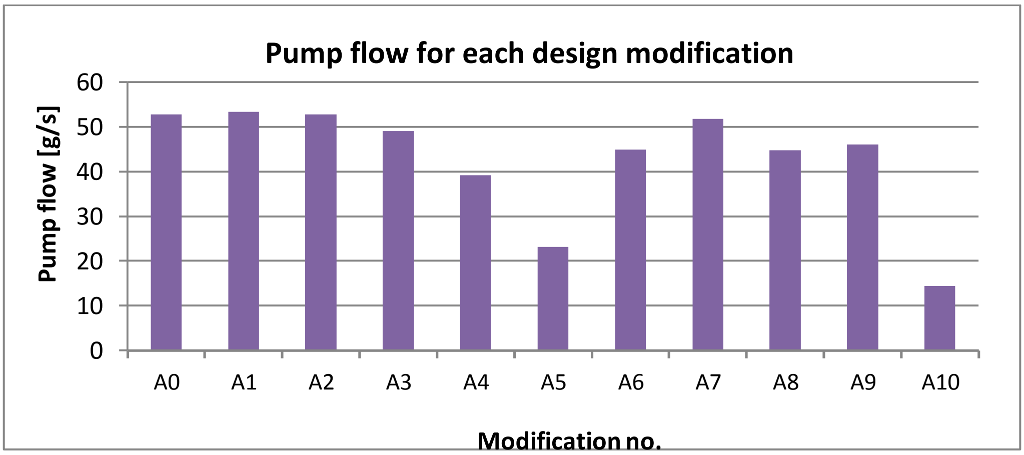


Figure 7.4: Bar schedule visualization pump flows of the different modifications

As can be seen in figure 7.4 above, the pump flow differs approximately likely the respectively vane heat flux but doesn't necessary mean they're linear to each other since the heat flux depend on more than just the mass flow rate.

It can also be seen that the brake shield sometimes contributes to an increased heat flux of the ventilations channels. For the nominal brake shield design, it could be seen that the opening of the brake shield at the vane inlets contributes to a venture-effect of the inlet pump flow. For the third brake shield modification, when the shield was displaced 5 mm from the disc (A3), this venturi-effect wasn't as clear since the surrounding pressure distribution had changed and the flow went in between the disc and shield instead of being directed into the ventilation channels, the brake shield could therefore work as a venturi-effect nozzle.

7.3 Former design studies

Pulugundla (2008) has done a comprehensive design study where he investigated how to optimize a curved vane design brake disc for maximum cooling performance in form of pump effect and convective cooling rate. He also investigates the heat rate uniformity because of the decreased risk of hot spot formation that comes with it. His study is limited since the application of his method demands the user that full vehicle studies, as in this report, has been done to make sure that a free rotation brake disc is representative to a full vehicle wheel house case.

He writes in his report that the best aero-thermal performances (pump flow and convective heat flux) he came up with are in the region of an inlet angles (β_1) between 35° and 45° while the outlet angles (β_2) between 50° and 60°, see figure 7.4 below. He also mention that this back swept blades have higher aero-thermal performance of the brake disc compared to the front swept blades because of their ability of causing separation of the flow at the inlets which lower the heat transfer.

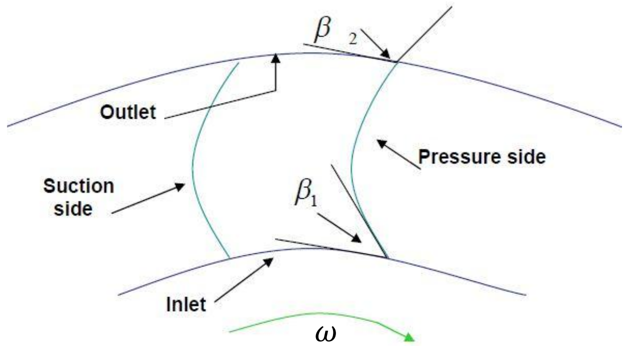


Figure 7.4: Schematic figure of inlet and outlet angles of a ventilation channel, taken from Pulugundla (2008).

Pulugundla (2008) also concludes the following about improved curve vane design for aero-thermal performance:

- With rotational speed of the brake disc comes improved aero-thermal performance due to the increase of heat transfer rate, mass flow rate and also heat transfer uniformity.
- Outlet angles of the cooling vanes have a bigger impact on the aero-thermal performance compared to the inlet angles.
- For higher temperatures of the brake disc, the aero-thermal performances decrease due to the decrease in heat transfer uniformity and also because of the decrease of density which comes with higher temperatures and further decrease in mass flow rate, which has been shown in this study

8 CONCLUSION AND DISCUSSION

The first conclusion of this study is that the pump flow behaviour of a rotating brake disc can be modelled according to earlier explanation with good correlation to physical tests, it's though notable that it only applies to straight vane design. This model is assumed to predict flow behaviour as good in a flow vehicle application where the flow is different from the free rotating case.

When applying the correlated brake disc model into the full mid-size vehicle model, it exhibits the same pump flow behaviour as the free rotating case. Considering the present covering wheel house design, not much pump flow interruption behaviour could be pointed out for the different cases tested. To gain some knowledge of the possibilities of using a free rotating brake disc case as a vane modification evaluating tool, two different designs were used in the study which showed good possibilities. For the specific case tested, it can be concluded that the simplified model can be used even though it shows an overrated relative improvement compared to the full vehicle comparison. The mid-size model used in this study is well covered when it comes to the front brake discs, so the disruption of pump flow is low. But for a bigger vehicle segment like an SUV, probably would exhibit more wheel house flow (higher ground distance) and the flow behaviour inside and around the brake disc would maybe be different from this case. Considering the vane flow behaviour, it can be concluded from chapter 6.5.2 that the vanes represent more than 50% of the total convective brake disc heat flux which makes it even more important to consider well designed vanes for better heat dissipation rate.

Considering the limitation of finally only investigating the vane design and not the pin design, is also something to take into account. Since the pin design doesn't pump as well as the vane design (see mass flow rate comparison in chapter 3.2), the pump flow behaviour would probably be interrupted easier by external flow, and the same conclusion according to simplified evaluation models would therefore not be valid.

For the brake shield redesign investigation, it can be concluded that an increased flow towards the brake disc doesn't mean that the total convective heat flux increases significant. The external brake disc heat flux increases significant, but the vanes can't pump through air as good when the surrounding flow becomes too intense. According to the centrifugal impeller theory in 2.2.1, the low vane inlet pressure is sucking in the external flow for pump cooling which becomes tougher when the flow is less concentrated. The vane heat flux therefore decreases in most of the "open up" modified cases, and the increase of total disc heat flux doesn't become that major. Then question therefore becomes if it is worth these modifications when the risk of dirt uptake increases significantly while the heat flux doesn't increases as much as expected. The conclusion therefore becomes that the wheel house modification must for each case be done in a strategic way where both the vane and external heat flux benefit from the increased flow while simultaneously a contamination study is evaluated.

8.1 Concluding summary

The final conclusions of the study could be summarized into following bullets:

- The aero-thermal flow behaviour of a ventilated brake disc can be well-predicted with a CFD model which has been correlated with experimental tests.
- For nominal wheel house designs of straight vane brake discs, the vane heat flux is the major total heat flux contributor.
- The brake shield opening close to the ventilations channel inlets is of great importance for the brake disc cooling.
- A venturi-effect of the pump flow for nominal brake shield design has been distinguished and works like an inlet pump flow nozzle.
- The brake disc cooling wasn’t improved very much when the brake shield was totally removed. It could therefore be concluded that an “open” design solution isn’t necessary a significant improvement. Strategic designs are needed to benefit both external and internal heat flux of brake disc.
- The closed rim design modification demonstrates the importance of having the desired “through-flow” of the wheel house for beneficial brake disc cooling.
- Because of the beneficial “through-flow” behavior for the vent-in design pump flow, the vent-out design doesn’t utilize this behavior, therefore the significant deterioration in brake disc cooling, mostly the vane heat flux.

8.2 Questions of issue

Finally, the questions of issue have to be answered to wrap up project into a reseal context by returning to the very first start of the project; the questions of issue.

Q1.

- *What are the basics of improved thermal management of a ventilated brake disc?*

The introduction chapter together with the theoretical part provided and summarized a comprehensive knowledge of the thermal management of a ventilated brake disc and pointed at the importance of convective cooling since it’s the major contributor in brake disc heat dissipating in a full vehicle application. Later on in the full vehicle investigation it was shown how important it is to consider aero-thermal ventilation channel designs since the major part of the convective cooling of the brake disc comes from the vanes for nominal designs of wheel house parts. The answer of the question would therefore basically be whole chapter 1 and 2.

Q2.

- *How can the aero-thermal flow behaviour of a ventilated brake disc be verified for simulations and analyses?*

After studying earlier similar literature and contacting experienced CFD employees at VCC and Chalmers, a couple of methods were tested for modelling and simulating the aero-thermal flow behaviour for correlation with the experimental investigation in the Volvo brake machine. The final model is explained in chapter 4 for more details about the model definitions and structure. It's also mentioned that the model is limited to e.g. straight vanes brake disc designs, not considering full thermal model etc. which should be considered before usage.

Q3.

- *How can a simplified evaluation model or method of a brake disc be developed and used, that represents a full vehicle aero-thermal flow behaviour?*

For certain brake disc modifications, the pump flow will decrease and so the aero-thermal vane performance due to the decreased vane gap. This relative case where the aero-thermal performances are compared both in free rotating case and in full vehicle case showed that a free rotating application can be used as a relative comparison considering aero-thermal vane performance. The improvements of the free rotating disc will though be overrated but the application itself is valid. See chapter 6 for more information. No full wheel house model was tested in this study which could be a simplified model of evaluating total brake disc aero-thermal performance, and not just evaluating the brake disc vanes. This is proposed more in depth in the future work chapter.

Q4.

- *What are the design directions considering improvement of the aero-thermal cooling performances of a ventilated brake disc?*

First, a comparison was made where the same brake disc was investigated but with a vent-in and a vent-out design in the full vehicle case. It could be seen that the preferred path of flow comes from the inside of the wheel house, underneath the car, through the wheel and out. This path is ideal for the vent-in design because the through-flow becomes more natural compared to the vent-out design where the pump-flow path has to go around the brake disc and shield to interfere with the out-flow to further flow into the vanes. This ineffectiveness shows in the aero-thermal results as well which can be seen in chapter 7. Also a couple of wheel house modifications was made and investigated in chapter 7 where it could be concluded that the total aero-thermal performance doesn't become significantly improved just by open up and increase the flow towards the brake disc. The major heat flux of the brake disc comes from the ventilation channels which mean that the flow modifications need to strategically benefit both the external and internal heat flux of the brake disc. More about this can be read in chapter 7 and 8.

9 FUTURE WORK

As for future studies, a lot more can be done in the area of brake cooling. An extension could be to look at a more general case compared to this. Many limitations have been described in this investigation as only one brake disc design type, constant velocity for full vehicle cases, constant temperature for full vehicle cases, the same model design for all cases (wheel design, rim design, front under hood design etc.). The conclusions drawn here should therefore also be investigated for all cases in order to draw a more general conclusion.

9.1 Thermal model

Another major limitation to this study is the fact that only convective thermal management is considered. A clear view of how the total heat flux is divided out on radiation, conduction and convection for different cases is still unclear, only guidelines of that knowledge are provided here. According to the literature study, guidelines of the importance of considering convective heat transfer has been clear and that the major part (>50%) of the total heat transfer of an average overall driving behaviour comes in form of convective heat transfer, but not more knowledge than that is clear. Therefore should a full vehicle thermal management model be developed e.g. by use of an external heat software like Radtherm, see appendix B1, where also correct heat distribution of a specific brake execution is defined over the brake disc, brake pads, caliper and other affected parts. This would also give a better understanding of the temperature distribution after a transient load case with cooling and therefore also predict and counteract eventual hot spot formation which is a common brake disc failure.

9.2 Design optimization

This project is a study that investigates flow behaviour of a brake disc both in full vehicle and free rotating cases and their correlation to each other. An extension could therefore be to optimize the brake disc design in order to dissipate heat in most effective way in a full vehicle case, which involves mass flow rate, total dissipated heat rate, cooling similarity along the brake disc surface etc. If for an example looking at an Audi Q7 front brake disc, it can be seen that it has been put in a lot of thoughts and development into the design stage. The ventilation channels are designed by over-lapping vanes with different lengths which make a pretty futuristic look, so a lot can be done just by investigating aero-thermal performance of competitors design strategies. Integrated with this could also be a design study of the vehicle redesign possibilities for even more brake disc cooling. This is a more comprehensive study though, since design modifications of the vehicle will probably interfere with other parts and functions of the vehicle. Take the example of the late shield modification chapter, chapter 7, if a new brake disc design would be set, a comprehensive contamination study would have to be integrated as well, even here could a competitor study be included.

9.3 Alternative studies

- This investigation only focuses on the front axle regarding wheel house flow cooling, so next step would be to investigate the flow behaviour of the rear axle.
- Bigger models usually don’t need the same cooling focus since the ground-vehicle distance allows a higher mass flow which partially provides cooling flow into the wheel houses, significantly more than mid-size and smaller cars do. A possible study could therefore be to investigate the need of cooling performance modifications among different car sizes and models.
- What are the environmental effects on the need of brake disc cooling? It would be interesting to investigate the needs of cooling performance at driving modes in the Alps compared to South Africa. Maybe is it possible to integrate a cooling performance design solution into an active safety system which activates on when needed?
- As explained earlier, the need of faster simulations and evaluating tools is ever increasing in today’s automotive industry since more and more CAE is replacing physical testing in a saving money and faster evaluating purpose. A future study could therefore be to investigate what’s really affecting the flow any noteworthy and if this could be simplified into a much smaller analysis domain (e.g. exporting and simulating only a wheelhouse and suspension with quarter of an underbody) compared to a full vehicle energy model. It has been seen and concluded in this study that the vane design can be evaluated by a free rotating brake disc by measuring the aero-thermal performance, even though the improvement is slightly overrated compared to the full vehicle improvement.
- If the simplified free rotating model is decided to be used as new vane design evaluation of aero-thermal performance, a tool could be developed for this. Only the CAD model should be necessary for importation and the whole model definitions and boundary conditions should be automated with a script in Ansa which further runs the volume mesh script in Harpoon and then runs the Fluent journal file. The tool should thus import the CAD file of the wanted brake disc and export all results like heat fluxes, mass flow rates etc. This tool could also be extended to any of the above mentioned suggested continuing investigations.
- Zhiling Qui, see Qui (2009), has investigated the flow behaviour effects of different rim designs and the study showed that depending on the specific rim design, the difference could be major. It has also been seen in this study that the rim design is of big importance for the aero-thermal performance of the brake disc, it would therefore be interesting to see how much different rim designs affect the wheel house through-flow and further the brake disc cooling performance.
- In today’s automotive industry, a lot of focus is laid on the fuel consumption performance of each model. It is therefore important to consider aerodynamic behaviour when executing major design changes. It would therefore be necessary and interesting to include a trade-off study between wheel house flow changes and the changes in aerodynamic drag and lift.

10 References

ASIRT (2013):

ASIRT, Association for Safe International Road Travel (organization that promotes global road safety).

http://www.asirt.org/KnowBeforeYouGo/RoadSafetyFacts/RoadCrashStatistics/tabid/213/Default.aspx (accessed 16th of January 2014).

Eggleston (2000):

Eggleston, D. (2000), Thermal Judder. EURAC Technical Bulletin 00034053, extracted from Doctoral thesis: An investigation into frictional surface interactions and their effect on cold judder.

Howstuffworks (2000):

Homepage of a scientific knowledge providing company.

http://auto.howstuffworks.com/auto-parts/brakes/brake-types/disc-brake1.htm (Accessed 17th of January 2014).

VolvoCars.com (2008):

Volvo Car Corporation homepage; news section.

https://www.media.volvocars.com/se/sv-se/media/pressreleases/17008 (accessed 29th of January 2014).

Motorera (2013):

Motor Era, dictionary of automotive term.

http://www.motorera.com/dictionary/di.htm (Accessed 17th of January 2014).

STARCS (2014):

STARCS, aerodynamics research and development.

http://www.starcs.se/pvt.aspx (Accessed 30th of January 2014).

Pulugundla (2008):

Puluguandla, G. (2008), Research thesis: CFD design analysis of ventilated disc brakes, Cranfield University of Engineering.

CFD-Online (2014)

CFD Online is an online centre for Computational Fluid Dynamics.

http://www.cfd-online.com/Wiki/Turbulence_modeling (Accessed 7th of Mars 2014).

FLUENT (2014)

Ansys Fluent online user's guide.

http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node568.htm (Accessed 18th of Mars 2014).

ANSYS (2014)

Ansys user's guide integrated in the software, version 14.5.0.

Incropera (2001)

Incropera, F.P. (2001). Introduction to Heat Transfer, 4th Edition. Wiley.

Thermopedia (2014)

Thermopedia is an online informational center which handles the physical phenomena of thermal dynamics.

http://www.thermopedia.com/content/660/ (Accessed 6th of May 2014)

Thermoanalytics (2014)

Thermoanalytics.com is the homepage of a growing company that develops different engineering software.

http://www.thermoanalytics.com/products/radtherm (Accessed 8th of May 2014)

Neys (2012)

Neys, A. (2012), Research thesis: In- vehicle brake system temperature model, Chalmers University of Technology, Gothenburg, Sweden, report no. 2012:38.

Qui (2009)

Qui, Zhiling. (2009), Research thesis: Wheel aerodynamic developments by module-based prototype rims and stationary rim shields, Chalmers University of Technology, Gothenburg, Sweden, report no. 2009:04.

APPENDIX A

A0

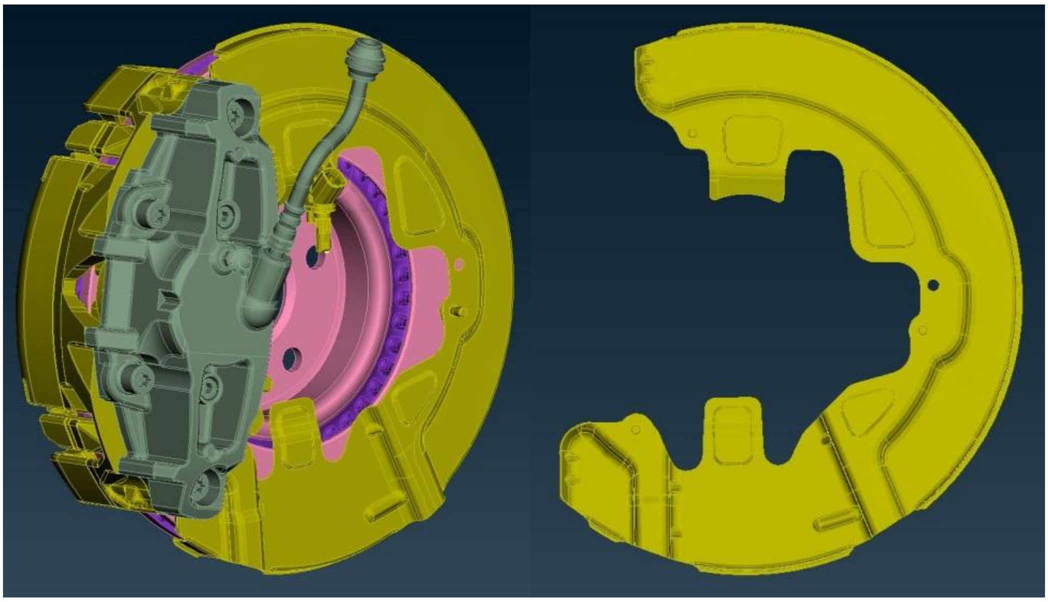


Figure A0: Visualization of the nominal brake shield model.

A1

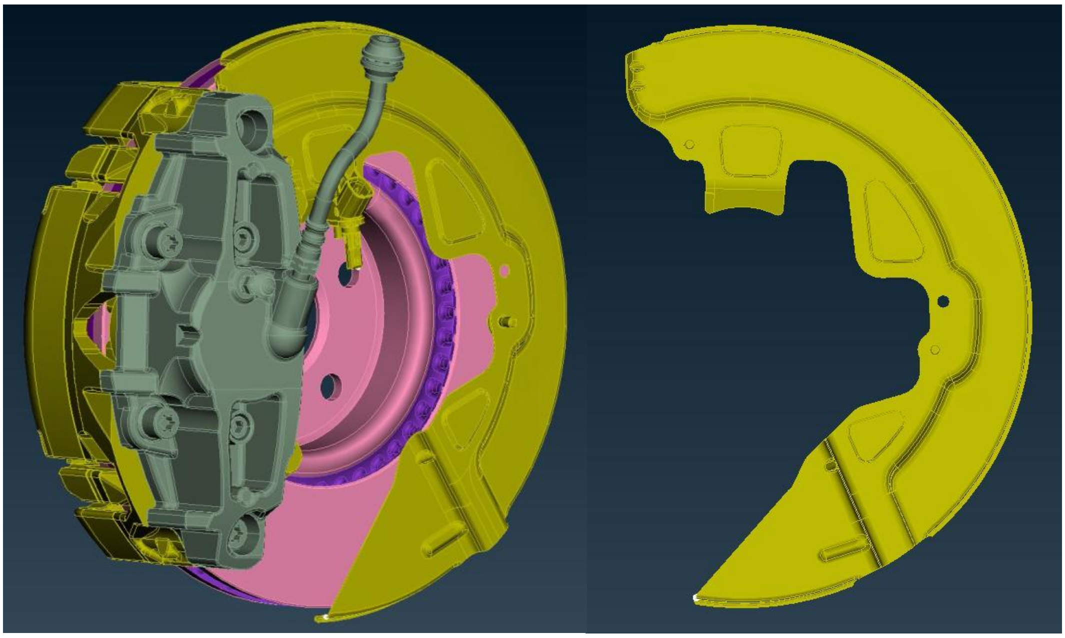


Figure A1: Visualization of the first brake shield modification; removed lower left part of brake shield.

A2

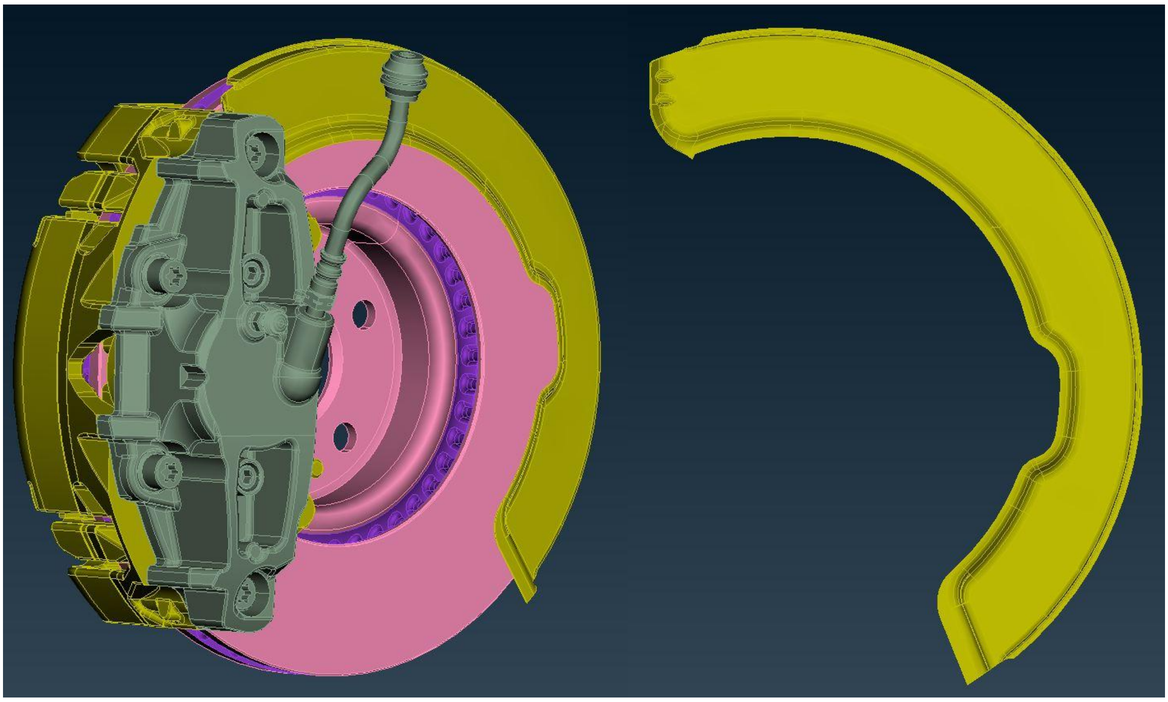


Figure A2: Second brake shield modification; removed lower part of brake shield thinner upper part.

A3

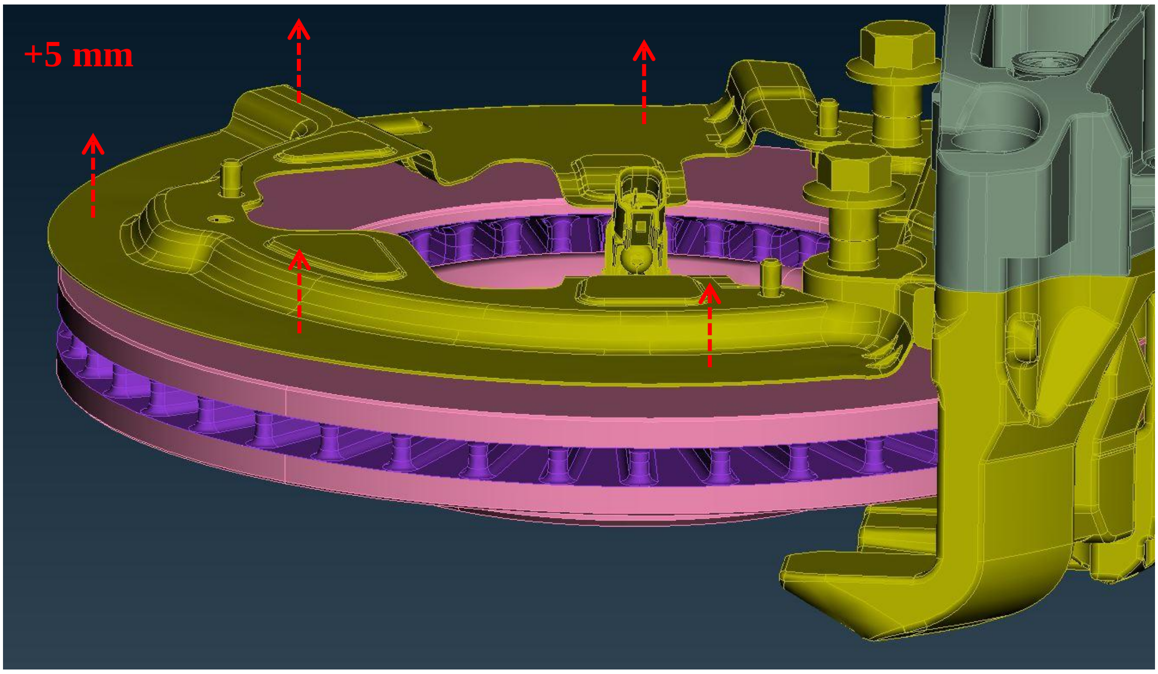


Figure A3: Third modification of brake shield; double the brake shield-disc distance and removed the outer brake shield lip.

A4

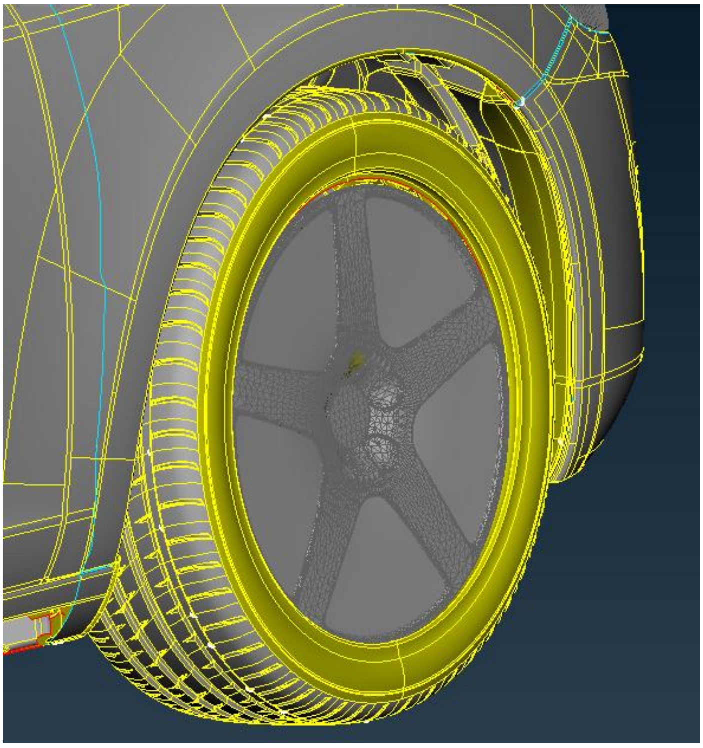


Figure A4: Modification of wheel house; totally covered outer rim.

A5

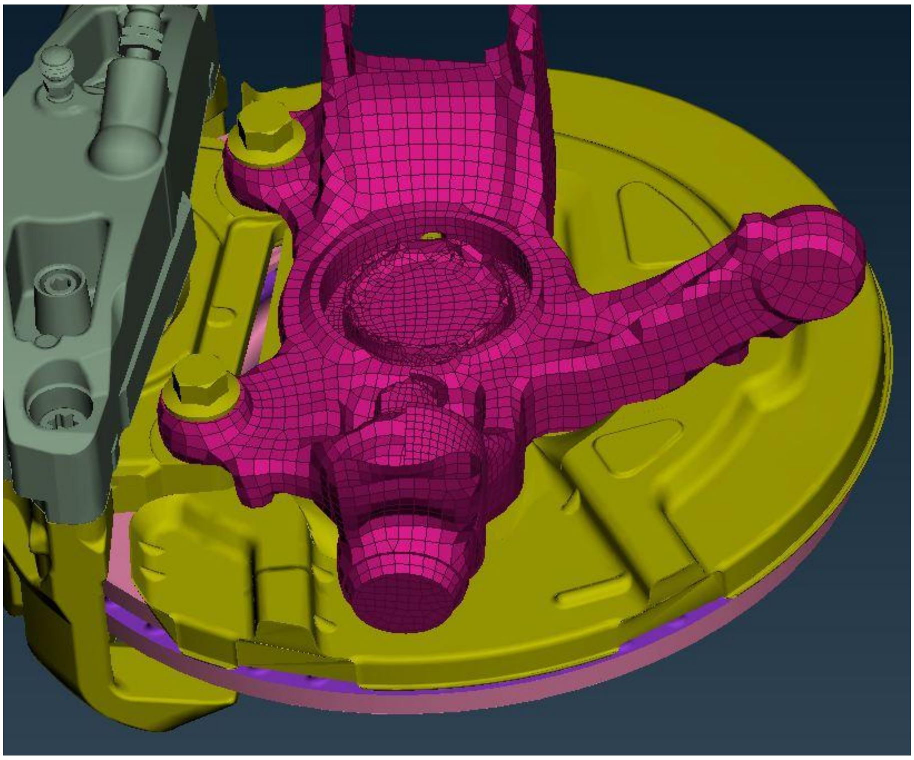


Figure A5: Fourth modification of brake shield, totally covered brake shield.

A6

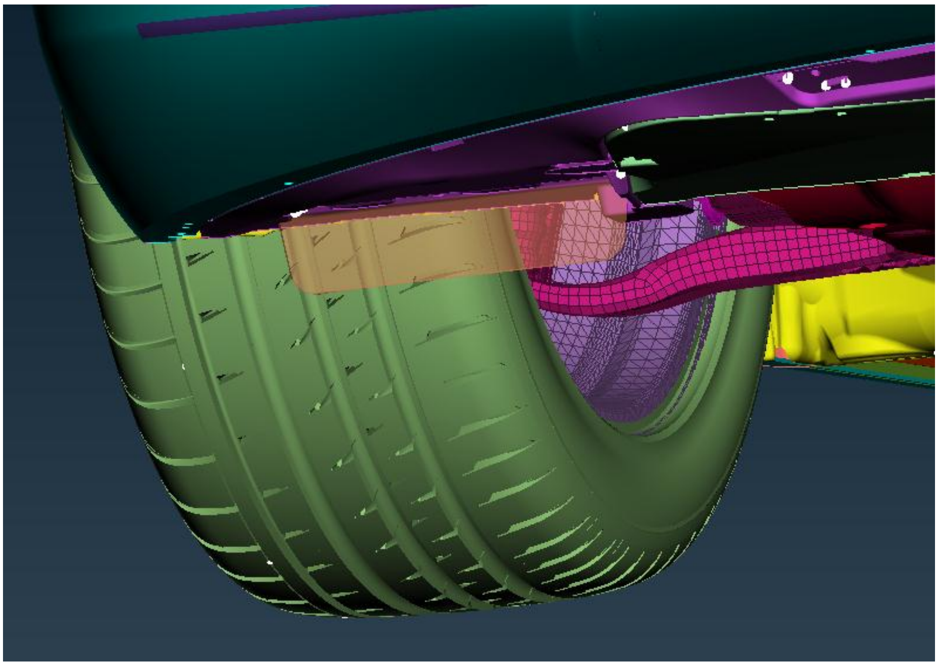


Figure A6: Modification of underbody; removal of the right front deflector (transparent orange flange).

A7

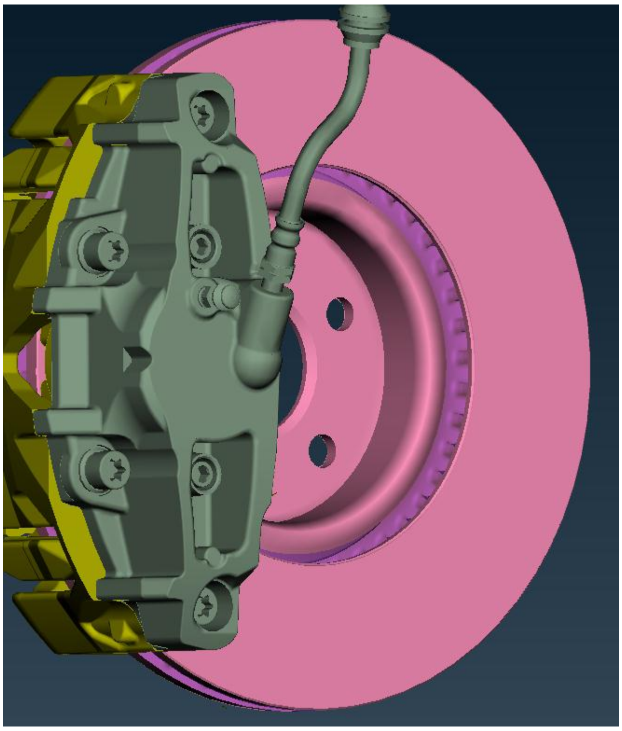


Figure A7: Fifth modification of brake shield; totally removed brake shield.

A9

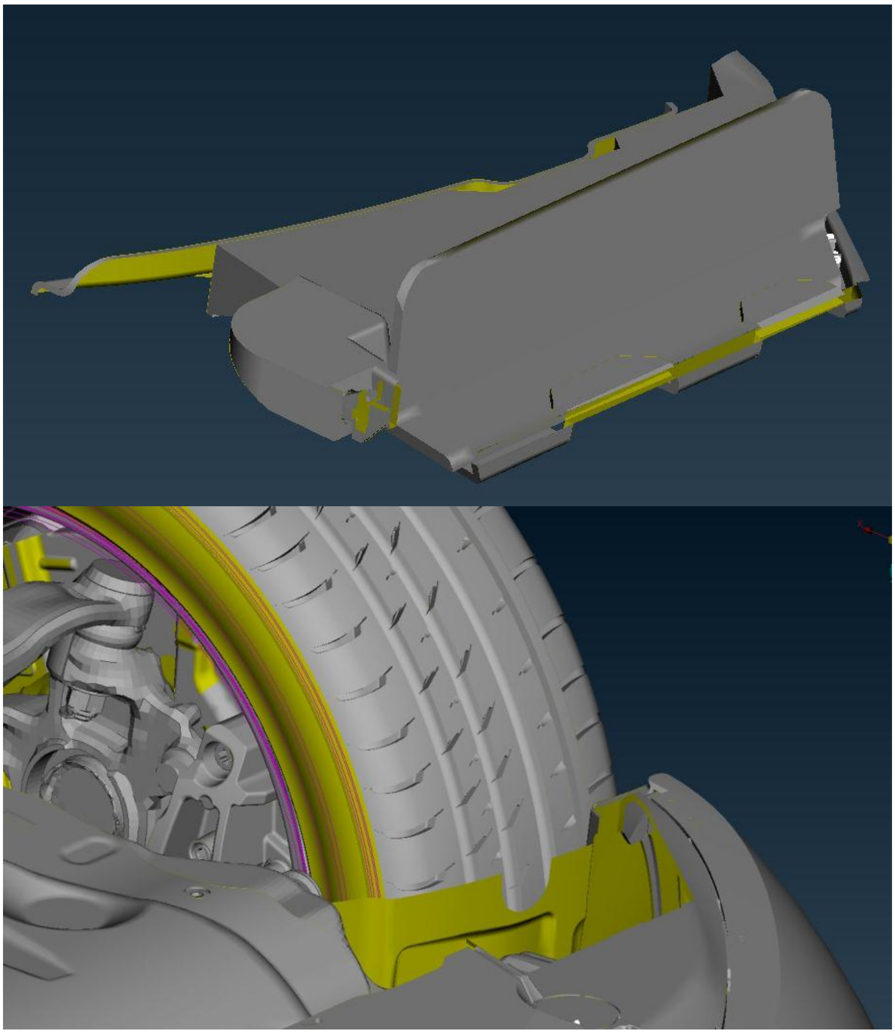


Figure A9: Modification of underbody; removed right front wheel deflector with attached surfaces to generate an opening for increased wheel house flow.

APPENDIX B

B0

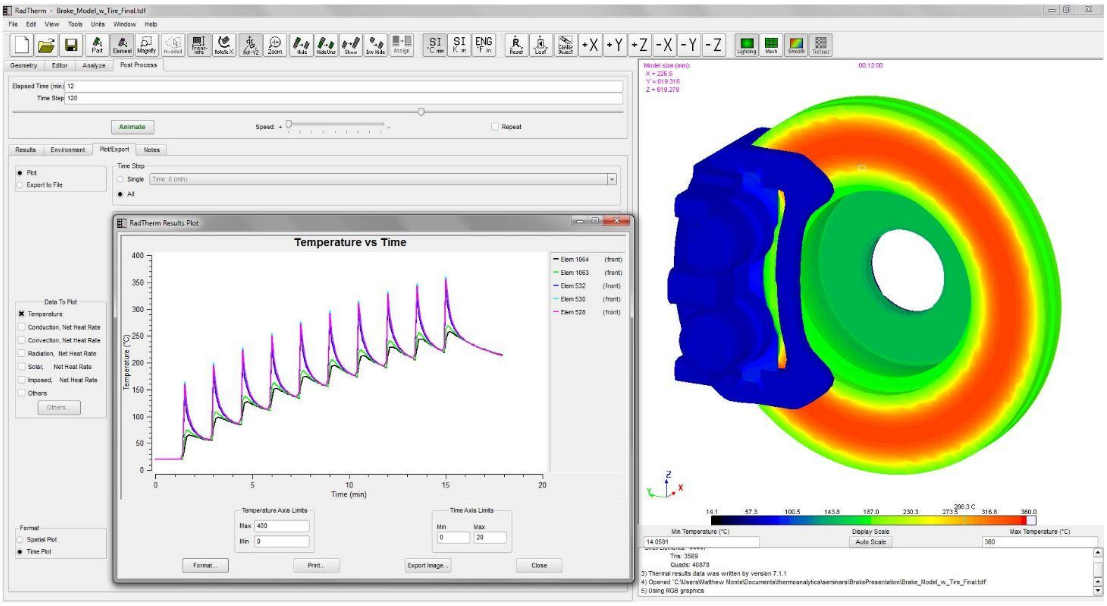


Figure B0: Example of the user interface of Radtherm.

APPENDIX C

This appendix chapter describes the different journal files used throughout the project including meshing and simulations.

C0

C0 described the journal file for the free rotating cases which is read by Fluent where it starts by reading the volume mesh from Harpoon, defining all analysis parameters, solving the case and finally writing the result file as both Fluent and Ensight versions.

```
;;############################################################################;;
;;######################### FLUENT RUN VERSION 14.5.0 #########################;;
;;############################################################################;;


;;---------------------------- DEFINE PARAMETERS ----------------------------;;

;; Disc temperature in Kelvin = 200 C
(define temp 473)

;; Rotational velocity in rpm = 115.74
(define rot 115.74)


;;------------------------------- BATCH OPTIONS ------------------------------;;
;; confirm file overwrite? no
;; exit on error? yes
;; hide questions? no
file/set-batch-options no yes yes


;;-------------------------------- READ MESH ---------------------------------;;

file/read-case ../HARPOON/harpoon_volmesh.msh


;;-------------------------------- SCALE GRID --------------------------------;;

;; x scale factor:
;; y scale factor:
;; z scale factor:
/grid/scale 0.001 0.001 0.001


;;---------------------------- ENABLE ENERGY MODEL ---------------------------;;

;; enable energy model? yes
;; compute viscous energy dissipation? no
;; incl pressure work in energy eq? no
;; incl kinetic energy in energy eq? no
;; incl diffusion at inlets? yes
/define/models energy yes no no no yes


;;-------------------------------- SET MODEL ---------------------------------;;

;; enable the realizable k-epsilon turbulence model? yes
/define/models/viscous/ke-realizable? yes


;;-------------------- USE GREEN-GAUSS CELL-BASED GRADIENTS -------------------;;

;; use green-gauss node-based gradients? no
;; use least squares cell based gradients? no
/solve/set/gradient-scheme  no no


;;--------------------------- NO CONVERGENCE CHECK ---------------------------;;

;; check for continuity/x-vel/y-vel/z-vel/energy/k/epsilon residuals? no
```

```
/solve/monitors/residual/check-convergence? no no no no no no no


;;--------------------------- SET SOLVER TO COUPLED ---------------------------;;

;; pressure velocity coupling scheme: 24
/solve/set/p-v-coupling 24

;; flow courant number: 20
;; explicit momentum under-relaxation: 0.25
;; explicit pressure under-relaxation: 0.25
/solve/set/p-v-controls 20 0.25 0.25

;; convective discretization scheme; for momentum: 1st order, for energy: 1st order.
/solve/set/discretization-scheme/mom 0
/solve/set/discretization-scheme/temperature 0


;;------------------------ AIR DEFINITION ------------------------;;
/define/materials/change-create air air yes ideal-gas no no yes sutherland three-
coefficient-method 1.716e-05 273.11 110.56 no no no


;;------------------------ VELOCITY INLET DEFINITION ------------------------;;

;; velocity specification method: magnitude and direction? no
;; velocity specification method: components? no
;; velocity specification method: magnitude, normal to boundary? yes
;; reference frame: absolute? yes
;; use profile for velocity magnitude? no
;; velocity magnitude: 0.0001
;; use profile for supersonic/initial gauge pressure? no
;; supersonic/initial gauge pressure: 0
;; use profile for temperature? no
;; temperature: 300
;; turbulent spec method: k-epsilon? no
;; turbulent spec method: intensity and length scale? yes
;; turbulent intensity [%]: 0.1
;; turbulent length scale: 0.22397
/define/boundary-conditions/velocity-inlet farfield_minx no no yes yes no 0.001 no 0
no 300 no yes 0.1 0.22397


;;---------------------- VELOCITY OUTLET DEFINITION ------------------------;;

;; use profile for gauge pressure? no
;; gauge pressure: 0
;; use profile for backflow total temperature? no
;; backflow total temperature: 300
;; backflow direction method: direction vector? no
;; backflow direction method: normal to boundary? no
;; backflow direction method: from neighboring cells? yes
;; turbulent spec method: k-epsilon? no
;; turbulent spec method: intensity and length scale? yes
;; backflow turbulent intensity [%]: 0.1
;; backflow turbulent length scale: 0.22397
;; radial pressure distribution equalibrium? no
;; specify average pressure specification? no
;; specify tageted mass flow rate? no
/define/boundary-conditions/pressure-outlet farfield_maxx no 0 no 300 no no yes no yes
0.1 0.22397 no no no


;;------------------ INTERNAL DISC SURFACE SETTINGS ------------------------;;

;; wall thickness: 0.002
;; use profile for heat generating rate? no
;; heat generation rate: 0
;; material name [aluminium], change? no
;; thermal BC type [heat-flux], change current value? yes
;; thermal bc type: temperature
;; use profile for temperature? no
;; temperature: 473
;; enable shell conduction? no
;; wall motion [motion-bc-stationary]: change current value? no
;; shear boundary condition [shaer-bc-noslip]: change current value? no
;; use profile for wall roughness height? no
```

```
;; wall roughness height: 0
;; use profile for wall roughness constant? no
;; wall roughness constant: 0.5
;; use profile for convectiv augmentation factor? no
;; convective augment factor: 1
/define/boundary-conditions wall wall-disc-01mm 0.002 no 0 no yes temperature no temp
no no no no 0 no 0.5 no 1


;; ------------------- ROTATING DISC SURFACE SETTINGS ---------------------;;

;; wall thickness: 0.002
;; use profile for heat generation rate? no
;; heat generation rate: 0
;; material name [aluminium], change? no
;; thermal bc type [heat flux], change? yes
;; thermal bc type: temperature
;; use profile for temperature? no
;; temperature: 473
;; enable shell conduction? no
;; wall motion [motion-bc-stationary], change? yes
;; wall motion: motion-bc-moving
;; shear boundary conditions [shear bc noslip], change? no
;; define wall motion relative to adjucant cell zone? no
;; apply a rotating velocity to this wall? yes
;; define wall velocity components? no
;; use profile for wall roughness? no
;; wall roughness height: 0
;; use profile for wall roughness constant? no
:: wall roughness constant: 0.5
;; rotational speed: -77.16
;; x-pos of rot axis: 1.702288116
;; y-pos of rot axis: 0.797434853
;; z-pos of rot axis: 0.464973217
;; x comp of rot direction: 0
;; y comp of rot direction: 1
;; z comp of rot direction: 0
;; use profile for convective argumentation factor? no
;; convective argumentation factor: 1
/define/boundary-conditions/wall wall-disc-rot-005mm 0.002 no 0 no yes temperature no
temp no yes motion-bc-moving no no yes no no 0 no 0.5 rot 1.702288116 0.797434853
0.464973217 0 1 0 no 1

/define/boundary-conditions/copy-bc wall-disc-rot-005mm wall-disc-rot-005mm*


;;----------------- ROTATING SHAFT SURFACE SETTINGS ---------------------;;

;; wall thickness: 0.002
;; use profile for heat generation rate? no
;; heat generation rate: 0
;; material name [aluminium], change? no
;; thermal bc type [heat flux], change? yes
;; thermal bc type: temperature
;; use profile for temperature? no
;; temperature: 473
;; enable shell conduction? no
;; wall motion [motion-bc-stationary], change? yes
;; wall motion: motion-bc-moving
;; shear boundary conditions [shear bc noslip], change? no
;; define wall motion relative to adjucant cell zone? no
;; apply a rotating velocity to this wall? yes
;; define wall velocity components? no
;; use profile for wall roughness? no
;; wall roughness height: 0
;; use profile for wall roughness constant? no
;; wall roughness constant: 0.5
;; rotational speed: -77.16
;; x-pos of rot axis: 1.702288116
;; y-pos of rot axis: 0.797434853
;; z-pos of rot axis: 0.464973217
;; x comp of rot direction: 0
;; y comp of rot direction: 1
;; z comp of rot direction: 0
;; use profile for convective argumentation factor? no
;; convective argumentation factor: 1
```

```
/define/boundary-conditions/wall wall-shaft-rot-02mm 0.002 no 0 no yes temperature no
temp no yes motion-bc-moving no no yes no no 0 no 0.5 rot 1.702288116 0.797434853
0.464973217 0 1 0 no 1

/define/boundary-conditions/copy-bc wall-shaft-rot-02mm wall-shaft-rot-02mm*


;; ------------------------ MRF SETTINGS ---------------------------;;

;; material name [air], change? no
;; specify source terms? no
;; specify fixed values? no
;; frame motion? yes
;; relative to cell zone -1
;; use profile for reference frame motion? no
;; reference fram speed? -77.16
;; use profile for reference frame x-velocity of zone? no
;; reference frame x-velocity of zone: 0
;; use profile for reference frame y-velocity of zone? no
;; reference frame y-velocity of zone: 0
;; use profile for reference frame z-velocity of zone? no
;; reference frame z-velocity of zone: 0
;; use profile for reference frame x-origion of rotational-axis? no
;; reference frame x-origin of rotational-axis: 1.702288116
;; use profile for reference frame y-origion of rotational-axis? no
;; reference frame y-origin of rotational-axis: 0.797434853
;; use profile for reference frame z-origion of rotational-axis? no
;; reference frame z-origin of rotational-axis: 0.464973217
;; use profile for reference frame x-component of rotational axis? no
;; reference frame x-component of of rotating axis: 0
;; use profile for reference frame y-component of rotational axis? no
;; reference frame y-component of of rotating axis: 1
;; use profile for reference frame z-component of rotational axis? no
;; reference frame z-component of of rotating axis: 0
;; reference frame user defined zone motion function: none
;; mesh motion? no
;; deactivated thread? no
;; laminar zone? no
;; porous zone? no
/define/boundary-conditions/fluid fluid-mrf no no no yes -1 no rot no 0 no 0 no 0 no
1.702288116 no 0.797434853 no 0.464973217 no 0 no 1 no 0 none no no no no

/solve/monitors/surface/set-monitor pump-flow "Area-Weighted Average" velocity-
magnitude 27 () no yes yes "pump-flow-velocity" 1
/solve/monitors/surface/set-monitor in-flow "Area-Weighted Average" velocity-magnitude
9 () no yes yes "in-flow-velocity" 1
/solve/monitors/surface/set-monitor out-flow "Area-Weighted Average" velocity-
magnitude 6 () no yes yes "out-flow-velocity" 1

;;----------------------- START ITERATIONS ----------------------------;;

/solve/init init
/solve/initialize/hyb-initialization
/solve/iterate 200
/solve/set/discretization-scheme/mom 1
/solve/iterate 2000


;;------------------------- WRITE DATA-CASE FILE ----------------------;;

file/write-case-data ../FLUENT/fluent_done.cas
file/export/ensight-gold ../ENSIGHT/fluent_done heat-flux heat-transfer-coef heat-
transfer-coef-wall turb-kinetic-energy velocity-magnitude tangetial-velocity pressure
pressure-coefficient specific-heat-cp temperature wall-temp-out-surf () yes * () * ()
no


;;------------- WRITE ITERATION-DATA FILE FOR CONVERGENCE CHECK --------;;

/report/surface-integrals/area-weighted-avg 27 () velocity-magnitude yes velocity.srp
/report/surface-integrals/area-weighted-avg 12 10 () heat-flux yes heat-flux.srp
/report/fluxes/heat-transfer yes yes heat-transfer-rate.flp


;;----------------------------- EXIT FLUENT --------------------------;;
exit yes
```

C1

C1 describes the Harpoon run file which read the surface mesh from Ansa, defines all mesh properties and finally writes the volume mesh.

```
import tgrid ../HARPOON/surf.msh
baselev 32
farfield global
farfield xmin -5782
farfield ymin -150
farfield zmin -2027
farfield xmax 7061
farfield ymax 1778
farfield zmax 2968
**refine       Start kommando
**0 3          Position 1: Box typ, 0=rätblock, Position 2: Upplösning på mesh 3 = 4
mm (med baselevel 32)
** 700 -450 350       Koordinat: xmin, ymin, zmin
** 1250 450 950       Koordinat: xmax, ymax, zmax
** refine **COOLPACK**
** 0 3
** 700 -450 350
** 1250 450 950
** refine ** HOOD
** 0 2
** 577 -950 264
** 2100 950 1100
** refine
** 0 0 ** AROUND CAR
** -500 -1500 0
** 6000 1500 2000
** refine ** WINDTUNNEL
** 0 -1
** -4000 -2500 0
** 10000 2500 2500

;;----------------------------- GROWTH RATE DEFINITION --------------------------;;

type hex
expand slow
mesh external
volume -3
remove
level 1
gminlev 1
gmaxlev 5
plevel *005mm 7 7 0
plevel *01mm 6 6 0
plevel *02mm 5 5 0
plevel *04mm 4 4 0
plevel *08mm 3 3 0
plevel *16mm 2 2 0
**plevel if* 5 7 0

;;------------------------- COORDINATE DEFINITIONS OF VOLUMES --------------------;;

vptkeep 1524 865.6 554 ** YTTRE FLUID
vptkeep 1630.48 804.4 577.47 ** KYLPAKETET
vptrename 1524 865.6 554 fluid-wt
vptrename 1630.48 804.4 577.47 fluid-mrf
smooth 2 0.98
smooth 2 all
smooth 2 0.98
** SET BOUNDARY CONDITIONS ALL FAN TO radiator FOR MEASUREMENT
setbc fan-* radiator
setbc farfield_maxx pressure-outlet
setbc farfield_minx velocity-inlet
setbc farfield_maxy symmetry
setbc farfield_miny symmetry
setbc farfield_maxz symmetry
save harpoon ../HARPOON/harpoon_volmesh
export fluent vol ../HARPOON/harpoon_volmesh.msh
```

C2

C2 describes the journal run script of the non-energy full vehicle model.

```
;; DEFINE VARIABLES
;; Define 1 if valid and 0 if not
(define OPEN_FRONT 1) ;; Set to 1 for open/closed cooling and 0 for closed front
(define RIGHT_SIDE 1) ;; Set to 1 for full car simulation and O for symmetry
;; FLUID PROPERTIES
(define DENSITY 1.205)
(define VISCOSITY 1.805e-05)
(define VELOCITY 27.77778)
;; GROUND POSITION
(define GROUND 0.173)
;; WHEEL POSITION AND ROTATIONAL VELOCITY
(define FW_X 1.708)
(define FW_Z 0.479)
(define RW_X 4.482)
(define RW_Z 0.479)
(define FW_OMEGA -90.5)
(define RW_OMEGA -89.9)
;; AXIS OF ROTATION (LEFT SIDE)
(define FW_AXIS_DX 0.00027)
(define FW_AXIS_DY 0.99982)
(define FW_AXIS_DZ -0.01891)
(define RW_AXIS_DX 0.00027)
(define RW_AXIS_DY 0.99934)
(define RW_AXIS_DZ -0.03636)
;; AXIS OF ROTATION (RIGHT SIDE)
(define FW_RS_AXIS_DX -0.00027)
(define FW_RS_AXIS_DZ 0.01891)
(define RW_RS_AXIS_DX -0.00027)
(define RW_RS_AXIS_DZ 0.03636)
;; CAR END (X VALUE)
(define CAR_END_X 5.401)
;; COOLING PACKAGE: NON-GEOMETRIC DATA - Data provided from THERMO
;; RADIATOR
(define RAD_INERTIAL_X 319.9)
(define RAD_INERTIAL_YZ 319900) ;; READ FROM BOUNDARY CONDITION'S FILE
(define RAD_VISCOUS_X 4.487e+07) ;; READ FROM BOUNDARY CONDITION'S FILE
(define RAD_VISCOUS_YZ 4.487e+10) ;; READ FROM BOUNDARY CONDITION'S FILE
;; CONDENSER
(define COND_INERTIAL_X 391.1) ;; READ FROM BOUNDARY CONDITION'S FILE
(define COND_INERTIAL_YZ 391100) ;; READ FROM BOUNDARY CONDITION'S FILE
(define COND_VISCOUS_X 3.99e+07) ;; READ FROM BOUNDARY CONDITION'S FILE
(define COND_VISCOUS_YZ 3.99e+10) ;; READ FROM BOUNDARY CONDITION'S FILE
;; CHARGE AIR COOLER
(define CAC_INERTIAL_X 398.7) ;; READ FROM BOUNDARY CONDITION'S FILE
(define CAC_INERTIAL_YZ 398700) ;; READ FROM BOUNDARY CONDITION'S FILE
(define CAC_VISCOUS_X 2.969e+07) ;; READ FROM BOUNDARY CONDITION'S FILE
(define CAC_VISCOUS_YZ 2.969e+10) ;; READ FROM BOUNDARY CONDITION'S FILE
;;
;; FAN_LARGE
(define FAN_LARGE_OMEGA 62.83) ;; THIS VALUE IS CONSTANT
;; COOLING PACKAGE: GEOMETRIC DATA
;; RADIATOR AXIS DIRECTION
(define RAD_LARGE_NORMAL_X 1)
(define RAD_LARGE_NORMAL_Y 0)
(define RAD_LARGE_NORMAL_Z 0)
;; CONDENSER AXIS DIRECTION
(define COND_LARGE_NORMAL_X 1)
(define COND_LARGE_NORMAL_Y 0)
(define COND_LARGE_NORMAL_Z 0)
;; CHARGE AIR COOLER AXIS DIRECTION
(define CAC_LARGE_NORMAL_X 1)
(define CAC_LARGE_NORMAL_Y 0)
(define CAC_LARGE_NORMAL_Z 0)
;; FAN AXIS DIRECTION
(define FAN_LARGE_NORMAL_X 0.99973)   ;; READ FROM MEASURE FILE
(define FAN_LARGE_NORMAL_Y 0.02237)   ;; READ FROM MEASURE FILE
(define FAN_LARGE_NORMAL_Z 0.00612)   ;; READ FROM MEASURE FILE
```

```
;; FAN LARGE CENTER
(define FAN_LARGE_X 1.124) ;; READ FROM MEASURE FILE
(define FAN_LARGE_Y -0.131) ;; READ FROM MEASURE FILE
(define FAN_LARGE_Z 0.675) ;; READ FROM MEASURE FILE
;;
;; FAN_SMALL
(define FAN_SMALL_OMEGA 62.83) ;; THIS VALUE IS CONSTANT
;; COOLING PACKAGE: GEOMETRIC DATA
;; RADIATOR AXIS DIRECTION
(define RAD_SMALL_NORMAL_X 1)
(define RAD_SMALL_NORMAL_Y 0)
(define RAD_SMALL_NORMAL_Z 0)
;; CONDENSER AXIS DIRECTION
(define COND_SMALL_NORMAL_X 1)
(define COND_SMALL_NORMAL_Y 0)
(define COND_SMALL_NORMAL_Z 0)
;; CHARGE AIR COOLER AXIS DIRECTION
(define CAC_SMALL_NORMAL_X 1)
(define CAC_SMALL_NORMAL_Y 0)
(define CAC_SMALL_NORMAL_Z 0)
;; FAN AXIS DIRECTION
(define FAN_SMALL_NORMAL_X 0.99986)   ;; READ FROM MEASURE FILE
(define FAN_SMALL_NORMAL_Y 0.01601)   ;; READ FROM MEASURE FILE
(define FAN_SMALL_NORMAL_Z 0.00441)   ;; READ FROM MEASURE FILE
;; FAN SMALL CENTER
(define FAN_SMALL_X 1.121) ;; READ FROM MEASURE FILE
(define FAN_SMALL_Y 0.199) ;; READ FROM MEASURE FILE
(define FAN_SMALL_Z 0.557) ;; READ FROM MEASURE FILE


;;-------------------- READ VOLUME MESH -------------------------;;

/file/set-batch-options yes yes no
rc ../HARPOON/harpoon_volmesh.msh


;;------------------------ RENAME ZONES -------------------------;;

/define/boundary-conditions zone-name farfield_maxx outlet
/define/boundary-conditions zone-name farfield_minx inlet
/define/boundary-conditions zone-name farfield_maxy symmetry1
/define/boundary-conditions zone-name farfield_maxz symmetry2
/define/boundary-conditions zone-name farfield_miny symmetry3
/define/boundary-conditions zone-name farfield_minz ground


;;----------------- SET ZONE BOUNDARY TYPES ----------------------;;

/define/boundary-conditions zone-type inlet velocity-inlet
/define/boundary-conditions zone-type outlet pressure-outlet
/define/boundary-conditions zone-type symmetry1 symmetry
/define/boundary-conditions zone-type symmetry2 symmetry
/define/boundary-conditions zone-type symmetry3 symmetry


;;------------- SET BC FOR SPECIFIC CONFIGURATIONS ---------------;;

;;/define/boundary-conditions zone-type baffle-underbody-rear-suspension-panel-4 wall
;;/define/boundary-conditions zone-type wall-coolpack-grilleshutter-upper-5 fan
;;/define/boundary-conditions zone-name wall-coolpack-grilleshutter-upper-5 fan-
coolpack-grilleshutter-upper-5


;;------------------------- VISCOUS MODEL ------------------------;;

/define/models/viscous/ke-realizable? yes


;;---------------------- MATERIAL PROPERTIES ---------------------;;

/define/materials/change-create air , yes constant DENSITY no no yes constant
VISCOSITY no no no


;;------------------------ MONITOR SETTINGS ----------------------;;
/solve/monitors/residual/check-convergence? no no no no no no
```

```
;; REFERENCE VALUES
/report/reference-values/density DENSITY
/report/reference-values/velocity VELOCITY
/report/reference-values/viscosity VISCOSITY
/report/reference-values/area 1
/report/reference-values/length 1


;;-------------------- PRINT MONITOR EVERY 5 ITERATIONS ---------------;;

/solve/set/reporting-interval 5


;;---------------------------- SOLVER SETTINGS -----------------------;;

(if (> OPEN_FRONT 0)
(begin
        (ti-menu-load-string(format #F "/solve/set/p-v-coupling 24"))
        (ti-menu-load-string(format #F "/solve/set/p-v-controls 20 .25 .25"))
        (ti-menu-load-string(format #F "/solve/set/discretization-scheme/mom 1"))
        (ti-menu-load-string(format #F "/solve/set/gradient-scheme no no"))
)
(begin
        (ti-menu-load-string(format #F "/solve/set/p-v-coupling 24"))
        (ti-menu-load-string(format #F "/solve/set/p-v-controls 20 .35 .35"))
        (ti-menu-load-string(format #F "/solve/set/discretization-scheme/mom 1"))
        (ti-menu-load-string(format #F "/solve/set/gradient-scheme no no"))
))
;;
/solve/set/gradient-scheme no no


;;--------------------- DEFINE INLET BOUNDARY CONDITION --------------;;

/define/boundary-conditions/velocity-inlet inlet n n y y n VELOCITY n 0 n n y 0.1 200


;;--------------------- DEFINE GROUND BOUNDARY CONDITION -------------;;

/define/boundary-conditions/wall ground yes motion-bc-moving no no no VELOCITY 1 0 0
no no 0 no 0.5

;; WHEELS

;;----------------- DEFINE WHEEL BOUNDARY CONDITION: LEFT SIDE -------;;

/define/boundary-conditions/wall wall-wheel-front-ls-tyre-omega-6 yes motion-bc-moving
no no yes no no 0 no 0.5 FW_OMEGA FW_X 0 FW_Z FW_AXIS_DX FW_AXIS_DY FW_AXIS_DZ
/define/boundary-conditions/wall wall-wheel-rear-ls-tyre-omega-6 yes motion-bc-moving
no no yes no no 0 no 0.5 RW_OMEGA RW_X 0 RW_Z RW_AXIS_DX RW_AXIS_DY RW_AXIS_DZ

;;------------------- COPY BC TO ALL MOVING PARTS ON LEFT SIDE -------;;

/define/boundary-conditions/copy-bc wall-wheel-front-ls-tyre-omega-6 wall-wheel-front-
ls-tyre-omega* wall-wheel-front-ls-rim-omega* wall-suspension-front-ls-brakedisc-
omega* ()
/define/boundary-conditions/copy-bc wall-wheel-rear-ls-tyre-omega-6 wall-wheel-rear-
ls-tyre-omega* wall-wheel-rear-ls-rim-omega* wall-suspension-rear-ls-brakedisc-omega*
()


;;------------------------- FLUID WHEEL MRF ZONES: LEFT SIDE ----------;;

/define/boundary-conditions/fluid fluid-wheel-front-ls-mrf no no no yes -1 n FW_OMEGA
n 0 n 0 n 0 n FW_X n 0 n FW_Z n FW_AXIS_DX n FW_AXIS_DY n FW_AXIS_DZ none n n n n
/define/boundary-conditions/fluid fluid-wheel-rear-ls-mrf no no no yes -1 n RW_OMEGA n
0 n 0 n 0 n RW_X n 0 n RW_Z n RW_AXIS_DX n RW_AXIS_DY n RW_AXIS_DZ none n n n n


;;------------------- DEFINE WHEEL BOUNDARY CONDITION: RIGHT SIDE -----;;
(if (> RIGHT_SIDE 0)
(begin
        (ti-menu-load-string(format #F "/define/boundary-conditions/wall wall-wheel-
front-rs-tyre-omega-6 yes motion-bc-moving no no yes no no 0 no 0.5 FW_OMEGA FW_X 0
FW_Z FW_RS_AXIS_DX FW_AXIS_DY FW_RS_AXIS_DZ "))
```

```
        (ti-menu-load-string(format #F "/define/boundary-conditions/wall wall-
suspension-front-rs-brakedisc-6 yes motion-bc-moving no no yes no no 0 no 0.5 FW_OMEGA
FW_X 0 FW_Z FW_RS_AXIS_DX FW_AXIS_DY FW_RS_AXIS_DZ "))
        (ti-menu-load-string(format #F "/define/boundary-conditions/wall wall-
suspension-front-rs-brakedisc-6.1 yes motion-bc-moving no no yes no no 0 no 0.5
FW_OMEGA FW_X 0 FW_Z FW_RS_AXIS_DX FW_AXIS_DY FW_RS_AXIS_DZ "))
        (ti-menu-load-string(format #F "/define/boundary-conditions/wall wall-wheel-
rear-rs-tyre-omega-6 yes motion-bc-moving no no yes no no 0 no 0.5 RW_OMEGA RW_X 0
RW_Z RW_RS_AXIS_DX RW_AXIS_DY RW_RS_AXIS_DZ "))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-wheel-
front-rs-mrf no no no yes -1 n FW_OMEGA n 0 n 0 n 0 n FW_X n 0 n FW_Z n FW_RS_AXIS_DX
n FW_AXIS_DY n FW_RS_AXIS_DZ none n n n n"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-disc-
front-rs-mrf no no no yes -1 n FW_OMEGA n 0 n 0 n 0 n FW_X n 0 n 0.4719 n
FW_RS_AXIS_DX n FW_AXIS_DY n FW_RS_AXIS_DZ none n n n n"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-wheel-
rear-rs-mrf no no no yes -1 n RW_OMEGA n 0 n 0 n 0 n RW_X n 0 n RW_Z n RW_RS_AXIS_DX n
RW_AXIS_DY n RW_RS_AXIS_DZ none n n n n"))
        ;; COPY BC TO ALL MOVING PARTS ON RIGHT SIDE
        (ti-menu-load-string(format #F "/define/boundary-conditions/copy-bc wall-wheel-
front-rs-tyre-omega-6 wall-wheel-front-rs-tyre-omega* wall-wheel-front-rs-rim-omega*
()"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/copy-bc wall-wheel-
rear-rs-tyre-omega-6 wall-wheel-rear-rs-tyre-omega* wall-wheel-rear-rs-rim-omega*
()"))
))


;;------------------------ FLUID COOLING PACK ZONES ------------------------;;

(if (> OPEN_FRONT 0)
(begin
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-rad no
no no no        no 0 no 0 no 0 no 0 no 0 no    1 no no no yes no no RAD_LARGE_NORMAL_X no
0 no 0 no 0 no 1 no RAD_LARGE_NORMAL_Z no no RAD_VISCOUS_X no RAD_VISCOUS_YZ no
RAD_VISCOUS_YZ yes no RAD_INERTIAL_X no RAD_INERTIAL_YZ no RAD_INERTIAL_YZ 0 0 no 1"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-cond no
no no no no 0 no 0 no 0 no 0 no 0 no 1 no no no yes no no COND_LARGE_NORMAL_X no 0 no
0 no 0 no 1 no COND_LARGE_NORMAL_Z no no     COND_VISCOUS_X no COND_VISCOUS_YZ no
COND_VISCOUS_YZ yes no COND_INERTIAL_X no COND_INERTIAL_YZ no COND_INERTIAL_YZ 0 0 no
1"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-cac no
no no no        no 0 no 0 no 0 no 0    no 0 no 1 no no no yes no no CAC_LARGE_NORMAL_X
no 0 no 0 no 0 no 1 no CAC_LARGE_NORMAL_Z no no CAC_VISCOUS_X no CAC_VISCOUS_YZ no
CAC_VISCOUS_YZ yes no CAC_INERTIAL_X no CAC_INERTIAL_YZ no CAC_INERTIAL_YZ 0 0 no 1"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-fan-
large-mrf no no no yes -1 no FAN_LARGE_OMEGA no 0 no 0 no 0 no FAN_LARGE_X no
FAN_LARGE_Y no FAN_LARGE_Z no FAN_LARGE_NORMAL_X no 0 no FAN_LARGE_NORMAL_Z"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/fluid fluid-fan-
small-mrf no no no yes -1 no FAN_SMALL_OMEGA no 0 no 0 no 0 no FAN_SMALL_X no
FAN_LARGE_Y no FAN_SMALL_Z no FAN_SMALL_NORMAL_X no 0 no FAN_SMALL_NORMAL_Z"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/wall wall-coolpack-
fan-small-shroud-stationary-6 yes motion-bc-moving no no no 0 1 0 0 no no 0 no 0.5"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/wall wall-coolpack-
fan-large-shroud-stationary-6 yes motion-bc-moving no no no 0 1 0 0 no no 0 no 0.5"))
        ;; COPY BC
        (ti-menu-load-string(format #F "/define/boundary-conditions/copy-bc wall-
coolpack-fan-large-shroud-stationary-6 wall-coolpack-fan-large-shroud-stationary-6*"))
        (ti-menu-load-string(format #F "/define/boundary-conditions/copy-bc wall-
coolpack-fan-small-shroud-stationary-6 wall-coolpack-fan-small-shroud-stationary-6*"))
))


;; BEGIN RUN.JOU: RUN SETTINGS

;; SCALE GRID TO METERS
/grid/scale 0.001 0.001 0.001
;; MESH QUALITY CONTROL
/mesh/size-info
/mesh/quality
/mesh/smooth-mesh
"quality based"
4
0.0005

/mesh/quality
;;
```

```
;; INITIALIZATION
/solve/initialize/initialize-flow
/solve/initialize/fmg-initialization/ yes

;; file/write-case-data ../FLUENT/fluent_done.cas
;; exit yes


;;---------------------- START OF ITERATION -------------------;;

/solve/iterate 2000

file/write-case-data ../FLUENT/fluent_done.cas


;;------------------------ MONITORS -----------------------;;

/solve/monitors/force/drag-coefficient y *fan* *wall* *baffle* () y n n n 1 0 0
/solve/monitors/force/lift-coefficient y *fan* *wall* *baffle* () y n n n 0 0 1
/solve/monitors/force/moment-coefficient y *fan* *wall* *baffle* () y n n n FW_X 0
GROUND 0 1 0
;;
;; ITERATING
;;
(if (> OPEN_FRONT 0)
(begin
       (ti-menu-load-string "/solve/iterate 5")
)
(begin
       (ti-menu-load-string "/solve/iterate 5")
))
;;

;;-------------------- REPORT FORCES ---------------------;;


/report/forces/wall-forces y 1 0 0 n
;;
;; REPORT MASS FLOW AND SURFACE INTEGRAL OF PTOT
(if (> OPEN_FRONT 0)
(begin
       (ti-menu-load-string "/report/fluxes/mass-flow n fan-coolpack-cac-out-6 fan-
coolpack-cond-out-6 fan-coolpack-rad-out-6 fan-exterior-front-grille-intake-5 fan-
exterior-front-spoiler-intake-5 , n")
       (ti-menu-load-string "/report/surface-integrals/integral fan-coolpack-cac-in-6
fan-coolpack-cac-out-6 fan-coolpack-cond-in-6 fan-coolpack-cond-out-6 fan-coolpack-
rad-in-6 fan-coolpack-rad-out-6 , total-pressure n")
       )
(begin
       (ti-menu-load-string "/report/fluxes/mass-flow n outlet , n")
))
;;
;; REPORT MOMENTS
;; MOMENT AROUND FRONT CONTACT POINT
/report/force/wall-moments no *fan* *wall* *baffle* , FW_X 0 GROUND 0 1 0 n
;; MOMENT AROUND REAR CONTACT POINT
/report/force/wall-moments no *fan* *wall* *baffle* , RW_X 0 GROUND 0 1 0 n
;;
;; FRONTAL AREA
/report/projected-surface-area (*wall*) 0.00005 1 0 0
;;
;; EXPORT ENSIGHT FILES
/file/export/ensight-gold
../ENSIGHT/ensi pressure total-pressure q yes (*) () no
/file/export/ensight-gold
../ENSIGHT/ensi_cell pressure total-pressure skin-friction-coef x-wall-shear z-wall-
shear x-face-area y-face-area z-face-area yplus q yes (*) () yes
;;
;; CDX PLOTS - COMPILE, LOAD, EXPORT
/define/user-defined/compiled-function compile "libudf" y
"../MACRO/write_face_forces.c" , ,
/define/user-defined/compiled-function compile "libudf2" y y
"../MACRO/write_face_forces_ext.c" , ,
/define/user-defined/compiled-function load "libudf"
/define/user-defined/compiled-function load "libudf2"
/file/read-macro ../MACRO/force@walls-export.scm
/file/read-macro ../MACRO/force@walls-export-ext.scm
```

```
(force@walls-export () "*")
(force@walls-export-ext () "*exterior*")
;;
;; EXPORT PLANE FOR POLAR PLOT
/surface/plane-point-n-normal Xplane1 CAR_END_X 0 0 1 0 0
/file/export/ascii ../FLUENT/polar_plot.txt Xplane1 () yes z-velocity y-velocity x-velocity pressure () no
;;


;;------------ EXPORT X AND Z-COORDINATE FOR THE COOLING PACKAGE ----------;;

/report/surface-integrals/vertex-avg fan-coolpack-cond-in-6 () x-coordinate no
/report/surface-integrals/vertex-avg fan-coolpack-cond-in-6 () z-coordinate no


;;-------------------------- CONTROL MASS FLOW PLANE -------------------------;;

/report/fluxes/mass-flow n fan-underbody-tunnel-4 , n
;;
;; CHECK MAX VELOCITY IN THE DOMAIN
/report/volume-integrals max * () velocity-magnitude no
/adapt/m-i-i-r y velocity 100 100000


;;------------------------------ WRTING RESULTS ----------------------------;;

;;
(if (> OPEN_FRONT 0)
(begin
       (ti-menu-load-string "wcd ../FLUENT/fluent-it3500.cas")
)
(begin
       (ti-menu-load-string "wcd ../FLUENT/fluent-it2000.cas")
))


;;--------------------- EXIT FLUENT -----------------------;;

exit yes yes yes
```

C3

C3 describes the additional journal file that enables the energy equation and first reads the first 2000 iteration case and enables all thermal definitions to the simulations and then iterates 3000 iterations more.

```
;; DEFINE VARIABLES
;; Define 1 if valid and 0 if not
(define OPEN_FRONT 1) ;; Set to 1 for open/closed cooling and 0 for closed front
(define RIGHT_SIDE 1) ;; Set to 1 for full car simulation and O for symmetry
;; FLUID PROPERTIES
(define DENSITY 1.205)
(define VISCOSITY 1.805e-05)
(define VELOCITY 27.77778)
;; GROUND POSITION
(define GROUND 0.173)
;; WHEEL POSITION AND ROTATIONAL VELOCITY
(define FW_X 1.708)
(define FW_Z 0.479)
(define RW_X 4.482)
(define RW_Z 0.479)
(define FW_OMEGA -90.5)
(define RW_OMEGA -89.9)
;; AXIS OF ROTATION (LEFT SIDE)
(define FW_AXIS_DX 0.00027)
(define FW_AXIS_DY 0.99982)
(define FW_AXIS_DZ -0.01891)
```

```
(define RW_AXIS_DX 0.00027)
(define RW_AXIS_DY 0.99934)
(define RW_AXIS_DZ -0.03636)
;; AXIS OF ROTATION (RIGHT SIDE)
(define FW_RS_AXIS_DX -0.00027)
(define FW_RS_AXIS_DZ 0.01891)
(define RW_RS_AXIS_DX -0.00027)
(define RW_RS_AXIS_DZ 0.03636)
;; CAR END (X VALUE)
(define CAR_END_X 5.401)
;; COOLING PACKAGE: NON-GEOMETRIC DATA - Data provided from THERMO
;; RADIATOR
(define RAD_INERTIAL_X 319.9)
(define RAD_INERTIAL_YZ 319900) ;; READ FROM BOUNDARY CONDITION'S FILE
(define RAD_VISCOUS_X 4.487e+07) ;; READ FROM BOUNDARY CONDITION'S FILE
(define RAD_VISCOUS_YZ 4.487e+10) ;; READ FROM BOUNDARY CONDITION'S FILE
;; CONDENSER
(define COND_INERTIAL_X 391.1) ;; READ FROM BOUNDARY CONDITION'S FILE
(define COND_INERTIAL_YZ 391100) ;; READ FROM BOUNDARY CONDITION'S FILE
(define COND_VISCOUS_X 3.99e+07) ;; READ FROM BOUNDARY CONDITION'S FILE
(define COND_VISCOUS_YZ 3.99e+10) ;; READ FROM BOUNDARY CONDITION'S FILE
;; CHARGE AIR COOLER
(define CAC_INERTIAL_X 398.7) ;; READ FROM BOUNDARY CONDITION'S FILE
(define CAC_INERTIAL_YZ 398700) ;; READ FROM BOUNDARY CONDITION'S FILE
(define CAC_VISCOUS_X 2.969e+07) ;; READ FROM BOUNDARY CONDITION'S FILE
(define CAC_VISCOUS_YZ 2.969e+10) ;; READ FROM BOUNDARY CONDITION'S FILE
;;
;; FAN_LARGE
(define FAN_LARGE_OMEGA 62.83) ;; THIS VALUE IS CONSTANT
;; COOLING PACKAGE: GEOMETRIC DATA
;; RADIATOR AXIS DIRECTION
(define RAD_LARGE_NORMAL_X 1)
(define RAD_LARGE_NORMAL_Y 0)
(define RAD_LARGE_NORMAL_Z 0)
;; CONDENSER AXIS DIRECTION
(define COND_LARGE_NORMAL_X 1)
(define COND_LARGE_NORMAL_Y 0)
(define COND_LARGE_NORMAL_Z 0)
;; CHARGE AIR COOLER AXIS DIRECTION
(define CAC_LARGE_NORMAL_X 1)
(define CAC_LARGE_NORMAL_Y 0)
(define CAC_LARGE_NORMAL_Z 0)
;; FAN AXIS DIRECTION
(define FAN_LARGE_NORMAL_X 0.99973)   ;; READ FROM MEASURE FILE
(define FAN_LARGE_NORMAL_Y 0.02237)   ;; READ FROM MEASURE FILE
(define FAN_LARGE_NORMAL_Z 0.00612)   ;; READ FROM MEASURE FILE
;; FAN LARGE CENTER
(define FAN_LARGE_X 1.124) ;; READ FROM MEASURE FILE
(define FAN_LARGE_Y -0.131) ;; READ FROM MEASURE FILE
(define FAN_LARGE_Z 0.675) ;; READ FROM MEASURE FILE
;;
;; FAN_SMALL
(define FAN_SMALL_OMEGA 62.83) ;; THIS VALUE IS CONSTANT
;; COOLING PACKAGE: GEOMETRIC DATA
;; RADIATOR AXIS DIRECTION
(define RAD_SMALL_NORMAL_X 1)
(define RAD_SMALL_NORMAL_Y 0)
(define RAD_SMALL_NORMAL_Z 0)
;; CONDENSER AXIS DIRECTION
(define COND_SMALL_NORMAL_X 1)
(define COND_SMALL_NORMAL_Y 0)
(define COND_SMALL_NORMAL_Z 0)
;; CHARGE AIR COOLER AXIS DIRECTION
(define CAC_SMALL_NORMAL_X 1)
(define CAC_SMALL_NORMAL_Y 0)
(define CAC_SMALL_NORMAL_Z 0)
;; FAN AXIS DIRECTION
(define FAN_SMALL_NORMAL_X 0.99986)   ;; READ FROM MEASURE FILE
(define FAN_SMALL_NORMAL_Y 0.01601)   ;; READ FROM MEASURE FILE
(define FAN_SMALL_NORMAL_Z 0.00441)   ;; READ FROM MEASURE FILE
;; FAN SMALL CENTER
(define FAN_SMALL_X 1.121) ;; READ FROM MEASURE FILE
(define FAN_SMALL_Y 0.199) ;; READ FROM MEASURE FILE
(define FAN_SMALL_Z 0.557) ;; READ FROM MEASURE FILE
;;

;; READ MESH
```

```
/file/set-batch-options yes yes no
rcd ../FLUENT/fluent_rot_done.cas


;; enable energy model? yes
;; compute viscous energy dissipation? no
;; incl pressure work in energy eq? no
;; incl kinetic energy in energy eq? no
;; incl diffusion at inlets? yes
/define/models energy yes no no no yes


;; convective discretization scheme; for momentum: 1st order, for energy: 1st order.
/solve/set/discretization-scheme/temperature 0

;; check for continuity/x-vel/y-vel/z-vel/energy/k/epsilon residuals? no
/solve/monitors/residual/check-convergence? no no no no no no no

/solve/set/under-relaxation turb-viscosity 0.5
/solve/set/under-relaxation temperature 0.5

;;----------------------- AIR DEFINITION -----------------------;;
/define/materials/change-create air air yes ideal-gas no no yes sutherland three-
coefficient-method 1.716e-05 273.11 110.56 no no no

;;----------------- INTERNAL DISC SURFACE SETTINGS -----------------------;;

;; wall thickness: 0.002
;; use profile for heat generating rate? no
;; heat generation rate: 0
;; material name [aluminium], change? no
;; thermal BC type [heat-flux], change current value? yes
;; thermal bc type: temperature
;; use profile for temperature? no
;; temperature: 473
;; enable shell conduction? no
;; wall motion [motion-bc-stationary]: change current value? no
;; shear boundary condition [shaer-bc-noslip]: change current value? no
;; use profile for wall roughness height? no
;; wall roughness height: 0
;; use profile for wall roughness constant? no
;; wall roughness constant: 0.5
;; use profile for convectiv augmentation factor? no
;; convective augment factor: 1

/define/boundary-conditions wall wall-suspension-front-rs-brakedisc-6 0.002 no 0 no
yes temperature no 473 no no no no



/define/boundary-conditions wall wall-suspension-front-rs-brakedisc-6.1 0.002 no 0 no
yes temperature no 473 no no no no



/define/boundary-conditions wall wall-mrf-7 0.002 no 0 no yes temperature no 473 no no
no no 0 no 0.5 no 1

;; MONITORS
;; /solve/monitors/force/drag-coefficient y *fan* *wall* *baffle* () y n n n 1 0 0
;; /solve/monitors/force/lift-coefficient y *fan* *wall* *baffle* () y n n n 0 0 1
;; /solve/monitors/force/moment-coefficient y *fan* *wall* *baffle* () y n n n FW_X 0
GROUND 0 1 0
;;
;; ITERATING
/solve/set/discretization-scheme/mom 0
/solve/set/discretization-scheme/temperature 0
/solve/iterate 3000

file/write-case-data ../FLUENT/fluent_energy_done.cas
exit yes


;; EXIT
exit yes yes yes
```

C4

C4 describes the mesh journal file that read the full vehicle surface mesh model from Ansa and then runs the volume mesh generation in Harpoon.

```
import tgrid ../HARPOON/surf.msh
**VERSION v4.4(a)**
**PREFERENCES USED**
**Max Skew 0.999500**
**Target Skew 0.980000**
**Max Face Warpage 40.000000**
**noreset**
**intersect**
**Separation Angle 40.0**
**Setting No. of Cells Between walls to 3**
**Setting BDF Exports to Short Format**
**Setting BDF Pyramid Treatment to use degenerate PENTA elements**
**Setting Max No. Separate Volumes to 100**
**Setting No. Cells for Auto Volume Delete to 5**
**Setting Part Description to use STL name**
**Setting Fluent Thin Wall Treatment to Single Sided**
baselev 40.000000
farfield global
farfield xmin -14232
farfield ymin -4750
farfield zmin 173
farfield xmax 35768
farfield ymax 4750
farfield zmax 10173
wlevel xmax -3
wlevel xmin -3
wlevel ymax -3
wlevel ymin -3
wlevel zmax -3
wlevel zmin -3
**REFINEMENT**
refine
0 0
-159 -1540 173
10034 1540 2061
refine
0 1
4482 -1110 173
7718 1110 1822
refine
0 2
618 -1110 173
5601 1110 867
refine
0 2
4482 -1110 173
5601 1110 1822
**MIRROR BOX**
refine
0 2
2193 -1177 960
4482 1177 1360
** REFINEMENT BOX FOR THE COND & RAD **
refine
2 5
1024 345 530
1024 -345 530
1100 -345 530
1100 345 530
1024 345 885
1024 -345 885
1100 -345 885
1100 345 885
** REFINEMENT BOX FOR THE CAC **
```

```
refine
2 5
950 345 392
950 -340 392
1100 -340 392
1100 345 392
950 345 545
950 -340 545
1100 -340 545
1100 345 545
**MESH METHODS**
type hex
expand slow
mesh both
remove
volume -3
**SINGLE LEVEL
level 1
gminlev 1
gmaxlev 5
plevel *-7 7 7 0
plevel *-6 6 6 0
plevel *-5 5 5 0
plevel *-4 4 4 0
pexp wall-mrf-7 4
pexp wall-exterior-front-a-pillar* 2
pexp wall-exterior-rear-5 2
pexp wall-exterior-rear-6 4
pexp wall-exterior-rear-spoiler-6 4
**************************************
**FIND VOLUMES**
***************************************
vfind start
baffle-underbody*
wall-engine-bay-*
wall-coolpack-fan-*
wall-powertrain-*
wall-suspension-*
wall-underbody-*
vfind end

****************************
**SORT OUT VOLUMES TO KEEP**
****************************
**MRF DISC FRONT RS**
vnamekeep begin fluid-disc-front-rs-mrf
wall-mrf-7
fan-mrf-inlet-7
fan-mrf-outlet-7
vnamekeep end
**MRF WHEEL FRONT LS**
vnamekeep begin fluid-wheel-front-ls-mrf
fan-wheel-front-ls-mrf-5
wall-wheel-front-ls-rim-stationary-5
vnamekeep end
**MRF WHEEL REAR LS**
vnamekeep begin fluid-wheel-rear-ls-mrf
fan-wheel-rear-ls-mrf-5
wall-wheel-rear-ls-rim-stationary-5
vnamekeep end
**MRF WHEEL FRONT RS**
vnamekeep begin fluid-wheel-front-rs-mrf
fan-wheel-front-rs-mrf-5
wall-wheel-front-rs-rim-stationary-5
vnamekeep end
**MRF WHEEL REAR RS**
vnamekeep begin fluid-wheel-rear-rs-mrf
fan-wheel-rear-rs-mrf-5
wall-wheel-rear-rs-rim-stationary-5
vnamekeep end
** CAC **
vnamekeep begin fluid-cac
fan-coolpack-cac-in-6
fan-coolpack-cac-out-6
wall-coolpack-cac-6
vnamekeep end
** COND **
```

```
vnamekeep begin fluid-cond
fan-coolpack-cond-in-6
fan-coolpack-cond-out-6
wall-coolpack-cond-6
vnamekeep end
** RAD **
vnamekeep begin fluid-rad
fan-coolpack-rad-in-6
fan-coolpack-rad-out-6
wall-coolpack-rad-6
vnamekeep end
** FAN LARGE**
vnamekeep begin fluid-fan-large-mrf
fan-coolpack-fan-large-in-6
fan-coolpack-fan-large-out-6
wall-coolpack-fan-large-blade-6
wall-coolpack-fan-large-shroud-stationary-6
vnamekeep end
** FAN SMALL**
vnamekeep begin fluid-fan-small-mrf
fan-coolpack-fan-small-in-6
fan-coolpack-fan-small-out-6
wall-coolpack-fan-small-blade-6
wall-coolpack-fan-small-shroud-stationary-6
vnamekeep end
*************************************
**SET BC ON FAN SURFACES (RADIATOR)**
*************************************
setbc fan-* radiator
*****************************
**SMOOTH**
*****************************
smooth 2 0.98
smooth 2 all
smooth 2 0.98
*************************************
**EXPORT TO FLUENT **
*************************************
vischeck
export fluent vol ../HARPOON/harpoon_volmesh.msh
**save harpoon ../HARPOON/harp_vol
```