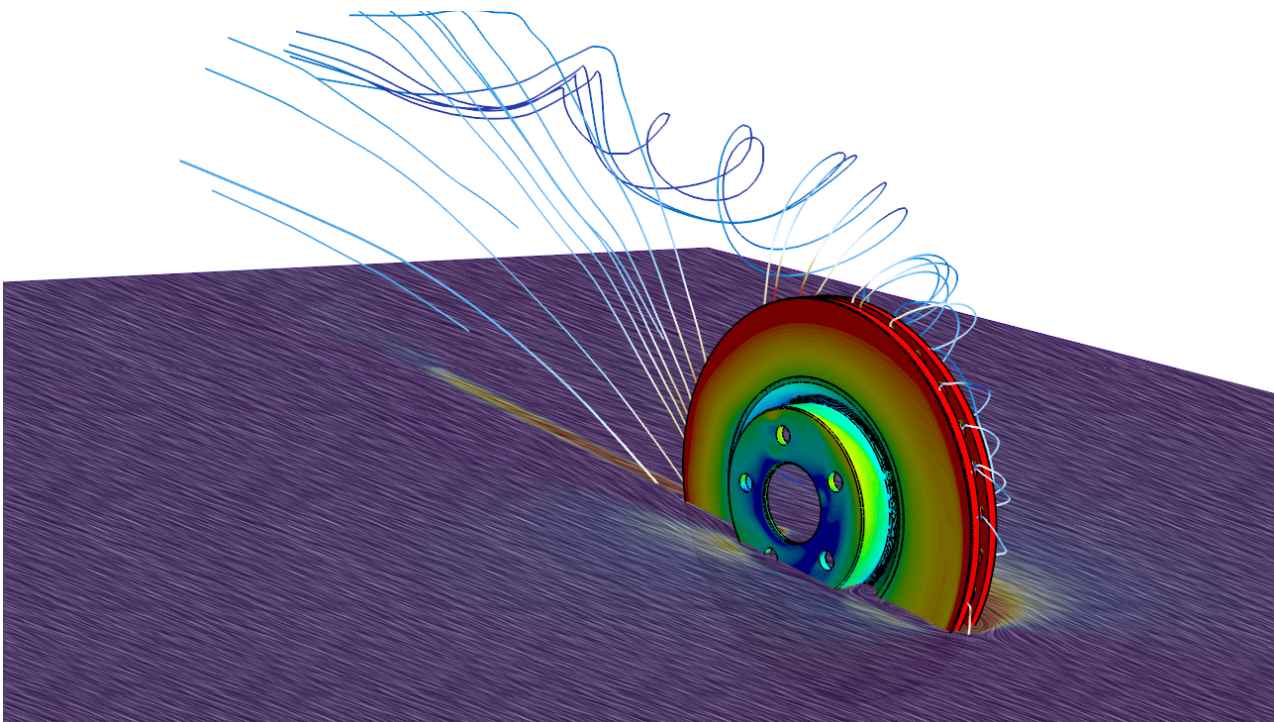




CHALMERS



Numerical Investigations of Brake Cooling Performance

Master's Thesis in Automotive Engineering

TIM DEKKER

VIGNESH KRISHNAN

MASTER'S THESIS IN AUTOMOTIVE ENGINEERING

Numerical Investigations of Brake Cooling Performance

TIM DEKKER
VIGNESH KRISHNAN

Department of Mechanics and Maritime Sciences
Division of Vehicle Engineering and Autonomous System
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2018

Numerical Investigations of Brake Cooling Performance
TIM DEKKER
VIGNESH KRISHNAN

© TIM DEKKER, VIGNESH KRISHNAN, 2018

Master's Thesis 2018:22
Department of Mechanics and Maritime Sciences
Division of Vehicle Engineering and Autonomous System
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Cover:

Vector plot of flow field around the disc with velocity streamlines travelling out through the disc. The disc surfaces are coloured with the Specified Temperature Heat Transfer Coefficient.

Chalmers Reproservice
Göteborg, Sweden 2018

Numerical Investigations of Brake Cooling Performance
Master's Thesis in Automotive Engineering
TIM DEKKER
VIGNESH KRISHNAN
Department of Mechanics and Maritime Sciences
Division of Vehicle Engineering and Autonomous System
Chalmers University of Technology

ABSTRACT

In the modern world, tough legislation on lowered emissions, leads the manufacturers to apply innovative strategies which involve aerodynamic improvements, such as covered rims. A covered rim is a good solution from an aerodynamics point of view, but poses serious constraints on the cooling performances of the brake discs, as it somewhat affects the cooling ability of the brake discs. To prevent critical situations that could lead to safety issues, such as decreased friction coefficients, brake hot-judder, increased wear, thermal cracking or even brake fluid boiling, the heat must be dissipated and hence, there is a demand for efficient cooling of brakes.

Traditionally, brake performance investigations were performed experimentally. However, with the computational power available today, these experiments can be simulated to save physical test time and resources. CAE simulations have shown good correlation with experimental results and can aid in incorporation of design changes at early stages of development. At Volvo Cars, these simulations are carried out using co-simulation where the aerodynamic and thermal solutions are calculated in parallel to get an estimate of the cooling performance.

This work examines the possibility to run mono-simulations using the CFD tool Star-CCM+ to test different approaches and investigate important parameters for brake disc cooling performance. During the project, investigations were carried out pertaining to:

- Various factors affecting the cooling performance
- Applicability of different Heat Transfer Coefficient definitions
- Effects of changes in brake disc design and rotation direction
- Influence of parts around the brake disc
- Approaches for brake cooling simulations using Star-CCM+

Some important observations made during the course of the project suggests that: the Virtual Local Heat Transfer Coefficient can be used for early comparison investigations which saves simulation time, the performance behavior due to rotational velocity variation can be predicted by linearization of the Heat Transfer Coefficient and there is an optimal point in variation of the design parameters where the best cooling performance of a brake disc type is achieved. This work was carried out at Chalmers University and with the support and valuable feedback from the brakes department at Volvo Cars.

Keywords: Brake cooling, CFD, heat transfer, vane designs, ventilated discs, rotation modelling, virtual local heat transfer coefficient

PREFACE

In this report, the results of the project work carried out during spring 2018 are reported. The work was carried out at Chalmers University of Technology in collaboration with the Brakes department at Volvo Cars Corp. The project aimed at developing a method to simulate the cooling performance of brake discs and also study the influence of external and geometric parameters on the cooling performance. The work was supervised by Gaël Le Gigan and Alessandro Travagliati at Volvo Cars and by Alexey Vdovin at Chalmers. This work was examined by Associate Professor Simone Sebben at the Vehicle Engineering and Autonomous System(VEAS) division at Chalmers University of Technology.

ACKNOWLEDGEMENTS

We would like to thank everybody involved directly or indirectly for their support and contribution that helped in the successful completion of this project. Our sincere thanks to our supervisors Gaël Le Gigan and Alessandro Travagliati at the Brakes department at Volvo Cars Corp., for sharing their experience and knowledge throughout the project. We would also like to thank Randi Franzke and Athanasios Tzanakis from Volvo Cars Corp., for contributing with strategies and a model, when we needed help.

Our sincere gratitude to our examiner, associate professor Simone Sebben for her invaluable input, guiding us through the technical know hows and for providing us with the information we needed.

We are highly indebted to our supervisor, Alexey Vdovin for spending all the precious time imparting his knowledge and expertise to us throughout the duration of the project. His continuous inputs have enhanced our thinking ability, the way we present our results and application of concepts. His support has motivated us to try and experiment different concepts.

It would not be fair on us if we did not acknowledge Emil Ljungskog and Magnus Urquhart for all their help and support, both with the workstations and with other technical inputs.

We would like to thank Chalmers University of Technology for letting us use the resources and providing licenses for softwares that aided in the successful completion of the project. We would like to extend our sincere gratitude to the members at VEAS division for making us feel home and comfortable during the course of the project work.

NOMENCLATURE

List of Abbreviations

CAD	Computer Aided Design
CAE	Computer Aided Engineering
CFD	Computational Fluid Dynamics
HT	Heat Transfer
HTC	Heat Transfer Coefficient
MRF	Moving Reference Frame
MW	Moving Wall
NS	Navier Stokes
RANS	Reynolds Averaged Navier Stokes
RBM	Rigid Body Motion

List of Symbols

a	acceleration [m/s^2]
α	closure coefficient in the $k - \omega$ turbulence model equation $[-]$
β	closure coefficient in the $k - \omega$ turbulence model equation $[-]$
β^*	closure coefficient in the $k - \omega$ turbulence model equation $[-]$
C	ratio of natural to forced convection $[-]$
C_n	concentration of species n [mol/m^3]
C_D	drag coefficient $[-]$
C_f	skin friction coefficient $[-]$
C_p	pressure coefficient $[-]$
$C_{p,f}$	specific heat [$J/kg \cdot K$]
δ	boundary layer thickness [m]
D_n	diffusion coefficient of species n [m^2/s]
ε	dissipation rate [$J/kg \cdot s$]
ϵ	emissivity [$J/kg \cdot s$]
F	force [N]
F_b	body force [N]
F_D	drag force [N]
g	gravitational constant [m/s^2]
G_b	turbulent kinetic energy due to buoyancy [J/kg]
G_f	grid flux [m^3/s]

h	energy [J]
H	heat transfer coefficient [$W/m^2 \cdot K$]
j_n	diffusional flux of species n [$mol/m^2 \cdot s$]
k	turbulent kinetic energy [J/kg]
κ	thermal conductivity [$W/m \cdot K$]
L	length [m]
m	mass [kg]
$\dot{m}$	mass flow rate [kg/s]
μ	dynamic viscosity of the fluid [$Pa \cdot s$]
∇	gradient operator $[-]$
Nu	Nusselt number $[-]$
ν	kinematic viscosity [$kg/m \cdot s$]
ω	turbulence dissipation [$1/s$]
P	pressure [Pa]
ϕ	scalar $[-]$
Pr	Prandtl number [Pa]
q	heat flux [W/m^2]
q_s	local surface heat flux [W/m^2]
Ra	Rayleigh number $[-]$
Re	Reynolds number $[-]$
ρ	density [kg/m^3]
S_h	source term in the energy equation $[-]$
σ	Stefan-Boltzmann constant [$W/m^2 \cdot K^4$]
σ_k	closure coefficient in the $k - \omega$ turbulence model equation $[-]$
σ_ω	closure coefficient in the $k - \omega$ turbulence model equation $[-]$
τ	shear stress [Pa]
θ	vane tangent [$^\circ$]
T	temperature [K]
t	time [s]
U	velocity [m/s]
u_*	friction velocity $[-]$
u, v, w	velocity components [m/s]
y^+	dimensionless wall distance $[-]$
y_c	height of near wall cell [m]

CONTENTS

Abstract	i
Preface	iii
Acknowledgements	iii
Nomenclature	v
Contents	vii
1 Introduction	1
1.1 Background	1
1.2 Purpose	1
1.3 Limitations	1
1.4 Specification of the Question	2
2 Theory	3
2.1 Vehicle Brake Systems	3
2.1.1 Brake System Assembly	3
2.1.2 Brake Discs	5
2.2 Governing Equations	6
2.2.1 Continuity Equation	6
2.2.2 Momentum Equation	6
2.2.3 Energy Equation	6
2.3 Turbulent Flow	7
2.3.1 RANS	8
2.3.2 URANS	8
2.3.3 IDDES	9
2.4 Heat Transfer	9
2.4.1 Convection	9
2.4.2 Conduction	9
2.4.3 Radiation	10
2.5 Heat Transfer Coefficients	10
2.5.1 Specified Temperature Heat Transfer Coefficient	10
2.5.2 Local Heat Transfer Coefficient	11
2.5.3 Specified y^+ Heat Transfer Coefficient	11
2.5.4 Virtual Local Heat Transfer Coefficient	11
2.6 Discretization	12
2.6.1 Mesh Independent Solution	13
2.7 Modeling of Rotation	13
2.7.1 Moving Wall	13
2.7.2 Moving Reference Frame	13
2.7.3 Rigid Body Motion	14
2.8 Field Functions	14
2.8.1 Pressure Coefficient	14
2.8.2 Skin Friction Coefficient	14
3 Methodology	15
3.1 Pre-processing	15
3.2 Simulation Set-up	16
3.2.1 Discretization	16
3.2.2 Solver Settings	17
3.2.3 Convergence	18
3.2.4 Investigations	18

3.3	Post-processing	23
4	Results and Discussions	24
4.1	Baseline Simulation Set-Up	24
4.1.1	Mesh Dependency Investigation	24
4.1.2	Rotation Modeling Techniques	25
4.1.3	Turbulence Models	27
4.2	Heat Transfer Coefficients	29
4.2.1	First Cell Height Dependent HTC	29
4.2.2	Non-First Cell Height Dependent HTC	29
4.2.3	Virtual Local HTC	30
4.2.4	Linearization and Scalability of HTC	32
4.3	Effect of Parameter Variations on Heat Transfer	34
4.3.1	Ambient Velocity	34
4.3.2	Disc Rotational Velocity	37
4.3.3	Ambient Temperature	38
4.3.4	Disc Temperature	39
4.4	Brake Disc Designs	40
4.4.1	Radial Vane Design	40
4.4.2	Tangential Vane Design	42
4.4.3	Curved Vane Design	44
4.5	Rotation Direction	45
4.6	Adjacent Parts	47
4.7	Gravity	51
4.8	Radiation	52
4.9	Solid Mesh Dependency Investigation	53
4.10	Cool-down Simulations	54
5	Conclusions	56
5.1	Future Scope	57
	References	59

1 Introduction

This master thesis is carried out as a part of the Masters program in Automotive Engineering at Chalmers University of Technology. The work was executed over a time period of 20 weeks.

The following report is structured into five chapters: Introduction, Theory, Methodology, Results and Discussions and Conclusions. The Introduction covers the background to explain the problem as well as the purpose and the specific questions that will be answered at the end of the project. Thereafter, a section covering the necessary theory merged with literature findings follows, for the reader to better understand the project. Then, the applied methodology and settings used are presented in the Methodology section. Based on the methodology, the findings are presented and discussed with regards to theory as well as literature findings in the Results and Discussion section. Lastly, the report ends with conclusions deduced from achieved results and discussions.

1.1 Background

One of the most critical features of a vehicle, is the ability to decelerate which is taken care of by the brake system. Brake systems can broadly be classified into friction, pumping or electromagnetic brake systems. The most commonly used brake system in the automotive industry is friction brakes. The brake system converts kinetic and potential energy into heat, which in turn is dissipated from the brake system into the surroundings. The heat transfer from the brake system can occur by three means i.e., conduction, convection and radiation. If the heat addition to the brake system is not dissipated at a high enough rate, the temperature could rapidly increase and in turn, result in overheating of brake components. This can result in brake fluid vaporization, reduced friction coefficient, brake judder, brake squeal, increased wear and thermal cracking. It then comes as no surprise, that it is important for vehicle manufacturers to investigate and thoroughly understand the thermal braking process to avoid these problems from occurring.

Traditionally, real world experiments were the primary method to investigate brake performance, although they are expensive and time-consuming. With the introduction of computer aided engineering (CAE), in particular computational fluid dynamics (CFD), these drawbacks can be overcome. Furthermore, CFD also makes it possible to visualize and perform investigations that are difficult to carry out in real life, it can be applied in early stages of development and is easily repeatable, yet again reducing cost and time.

Commonly, the computational investigations are carried out coupling two software, i.e. co-simulations; one software solving the heat transfer and the other software solving the aerodynamic flow [1, 2, 3]. For co-simulations, the boundary conditions play a key role since the two software exchange data. The reason being that, if erroneous boundary conditions are applied, an erroneous solution may be achieved. On the contrary, simulating both parts in one software may result in a higher modeling effort and larger models, thereby a larger computational cost, but can, when set up correctly, result in a more detailed solution [4].

1.2 Purpose

This project aims at defining a simulation methodology for brake cooling investigations, i.e. creating a mesh independent solution, recommend a rotation technique and turbulence model using a CFD solver, such as STAR-CCM+ which is used in this study. The established configuration will then be used to investigate heat transfer coefficients, the effect of adjacent parts on the flow field, brake disk designs, rotational direction and parameter dependencies on brake cooling performance.

1.3 Limitations

This work is performed as a master thesis by two students and is limited to a time period of 20 weeks. The main focus is on the brake discs and few adjacent parts, meaning that the influence of e.g. the wheelhouse, cooling duct or full car simulations will not be investigated. Furthermore the simulations will consider the aerodynamic and thermodynamic field, but do not cover structural analysis. Consequently, effects such as

warping, stress concentrations and cracks will not be examined in this work. The brake disc will have properties of a used, but not worn out or damaged disc. To verify the work, the obtained results and used models will be compared to literature and previous studies, but no real-world testing will be involved in the project.

1.4 Specification of the Question

Based on the outcome of the project, it should be possible to answer the following questions,

- How should the optimal configuration be defined for a limited computational resource?
- How should the Heat Transfer Coefficient be defined to reflect the heat transfer of the prescribed application?
- How does the flow field and cooling performance change with altered velocity and temperature?
- How does the flow field and cooling performance change for different brake disc designs, such as radial, tangential and curved vanes?
- How is the flow field and cooling performance affected by rotation direction?
- How does addition of adjacent parts, such as brake shield and brake calliper, affect the flow field and cooling performance?

2 Theory

The following section is meant to introduce the reader to how and what a typical brake systems consists of as well as to provide a brief theoretical introduction into computational fluid dynamics in order to give the reader a background within the field. The section also covers findings and relevant discussion from the literature survey.

2.1 Vehicle Brake Systems

Most common automotive brakes that are being used today are disc brakes, but some car manufacturers still produce light-weight vehicles with drum brakes, mainly on the rear. Both types of brakes, reduce the motion of the vehicle by using friction. Whilst a disc brake operates by changing the hydraulic pressure that actuates the piston(s) which pushes the pads against a rotating disc, drum brakes can be actuated by hydraulic or a pneumatic pressure wherein brake pads rub against a drum. Both of these types of brakes are exposed to high temperature during braking and hence, require optimum cooling.

The reason for car manufacturers to change from drum brakes to disc brakes is the increased power output of vehicles and the increased legislation on brake performance. Since drum brakes are encapsulated, they overheat more easily compared to disc brakes but are less exposed to dust. This, in combination with that brake discs are more reliable has resulted in a shift from drum brakes to disc brakes [5].

2.1.1 Brake System Assembly

A brake system consists of pedal assembly, brake booster, master cylinder, regulating valves and the foundation brakes. On the command of either the driver by pushing the brake pedal, or by the vehicle, if it is equipped with active safety systems, the pressure of the brake fluid is increased using the brake booster. To determine how much pressure should go to each calliper, and thereby how hard the piston(s) should be pressed against the brake pads and in turn against the brake disc, the master cylinder and regulating valves are used.

The foundation brakes include the brake system parts located at the wheel, i.e. brake disc, brake pads, brake calliper and the brake dust shield, as defined in Figure 2.1, and will be presented more into detail in the following subsections.

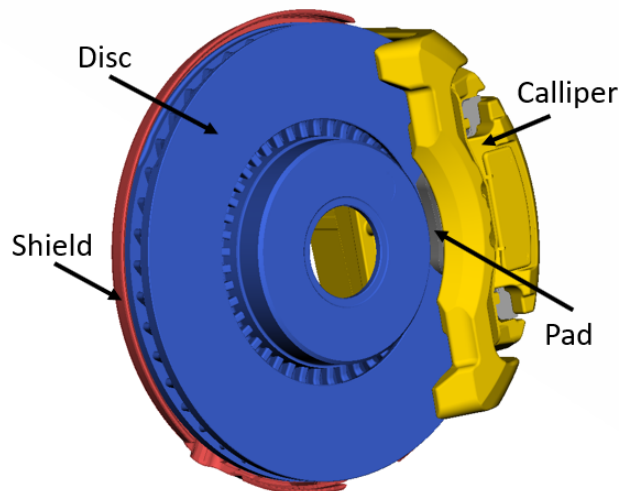


Figure 2.1: *The layout of a typical automotive disc brake.*

Brake Shield

The brake shield's (also known as the dust shield), main function is to protect the inner friction surface of the brake disc from debris and dirt, such as sand and mud. If dirt accumulates or gets trapped between the brake disc and the brake pads, the friction between the two could decrease, thereby decreasing the performance of the brake system. Another hazard is that dirt can damage the surfaces, thereby decreasing the performance and lifetime of the brake parts. However, despite the advantages of having a brake shield, the brake shield will influence the flow around the disc and hence, the heat transfer. The main disadvantage is that it will cover the inner friction surface and disturb the flow field and as found in literature, thereby lower the disc's cooling performance [6, 7, 8]. Additionally, the dust shield may introduce noise due to stones getting trapped between the brake disc and the dust shield.

Brake Pad

The brake pads are the parts being pushed against the friction surfaces in order to stop the brake discs and thereby the vehicle. Therefore it is imperative that the friction between the two is high and consistent, and that the pad can withstand high temperatures.

Commonly, brake pads were made with asbestos since this resulted in a high friction coefficient, and thereby high brake system performance. However, since asbestos is carcinogenic, it is not used in brake pads developed today. Generally, brake pads are made out of four components; reinforcing fibres, binders, fillets and frictional additives [9].

Brake Calliper

The brake callipers houses the piston(s), which pushes the brake pads against the brake discs due to the increase in the pressure of the brake fluid. Two calliper types that can be found in vehicles today are fixed callipers and floating callipers, also known as sliding callipers. A fixed calliper uses pistons on both sides, whilst a floating calliper uses a one or more pistons on only one side. When the floating calliper pushes the brake pad against the disc, the calliper will slide along the rotation axis of the disc, until the pad on the opposite side also is pushed against the brake disc. On passenger vehicles, floating callipers are more commonly applied since these are less expensive and occupy less volume around the brake disc. On the contrary, fixed callipers can apply a more evenly distributed clamping force right away and give a better brake feel. This is why fixed callipers are more commonly applied on performance and luxury vehicles.

Similar to the brake shield, the brake calliper will decrease the cooling performance of the brake disc since it interferes with the flow field and covers approximately 15-35 % of the brake disc. As an example of this, it has been shown that by moving the calliper from facing the flow to not facing the flow behind the axle, the heat transfer can be increased both for the brake disc and the calliper [8].

Brake Disc

Equally important to the brake pads, are the brake discs, which help in bringing the vehicle to a stand still. Previously, solid brake discs were used, but, due to the increased performance and high speeds that today's cars can reach, ventilated brake discs are becoming a first design choice. Ventilating, or vented, brake discs work in such a way that air is pumped through the brake disc due to a centrifugal force generated by rotation, similar to a centrifugal pump, improving the cooling capability of the brake disc. Many different materials have been and are in use, such as aluminum, cast steel, high carbon cast iron and even ceramic composite materials, although, grey cast iron is most frequently used for automotive brake discs.

Many different types of vented brake discs exist. Not only can the vanes be different, e.g. radial, tangential, curved, pillar or diamond vanes to mention a few, but, the surface of the disc can also vary, e.g. drilled or slotted/grooved disc. Studies have shown that the vane configuration and the brake disc parameters have a big impact on the cooling capacity of the brake disc [10].

2.1.2 Brake Discs

Three common type of brake disc designs are radial, tangential and curved vane design brake discs, as displayed in Figure 2.2. Radial vane design brake discs are the most simple and commonly used ventilated brake discs. They can be said to be non-directional, i.e. they can be mounted on either side of the vehicle and still perform equally well. The reason for their wide application is that they perform better than solid discs, are lighter compared to tangential and curved vane discs and are non-directional. However, it has been shown that when the vane angle is low, i.e. for radial vanes, a high mass flow through the disc is necessary and separation and recirculation may occur on the inlet of the vanes, resulting in less flow through the vanes [11].

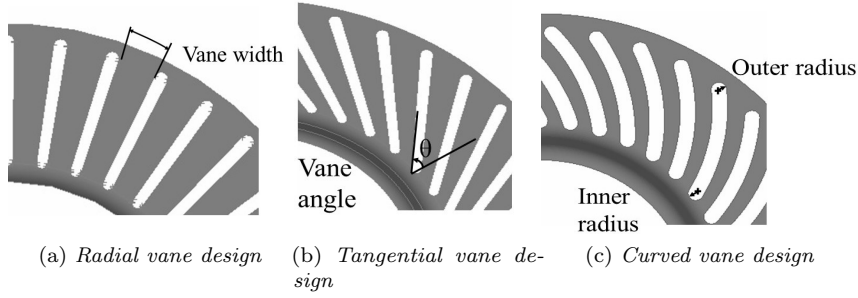


Figure 2.2: *Radial, Tangential and Curved vane type brake disc.*

The tangential vane design is similar to the radial vane design as they have straight vanes, but placed at an angle, defined by θ in Figure 2.2, and the curved vane type can be described as bent tangential vanes. Both these types of vanes are directional, meaning that the manufacturer either has to generate two different brake disc designs for the two sides of the vehicle, or has brakes that will perform better on one side of the vehicle. These brake discs generally have longer vanes compared to the vanes of radial vane design and perform better when rotated in the correct direction. Having a longer vane introduces more material and more cooling area, hence better interaction with the flow. A disadvantage is that the brake discs become heavier and will be more expensive to manufacture, especially if the manufacturer creates one disc for each side of the car.

To produce non-directional high performance brake discs, pillared and diamond vane type discs, as displayed in Figure 2.3, were introduced. These discs partly include the superior performance that curved vane discs have without having to take rotational direction into consideration. However, they come with the drawback of an increased manufacturing cost and casting scrapping rate due to the fact that the core is less stable. Lately, performance vehicle manufacturers have started mixing different patterns, e.g. combining curved with discontinuous curved vanes.



Figure 2.3: *Vane-layout for a mix pattern vane brake disc [12].*

The common parameters varied between brake discs are the vane width, vane angle (for tangential vane design brake discs), vane curvature (for curved vane designs brake discs) and the number of vanes, as shown in Figure 2.2. When the vane width is increased and the number of vanes are kept constant, the volume through which

the flow passes is increased. This often results in an increased mass flow resulting in an increase in heat transfer [13]. A similar effect, i.e. increased mass flow, can be found when the number of vanes are increased, often increasing the heat transfer [14]. As goes for the vane tangent angle, studied have shown that an altered angle has resulted in increased heat transfer performance in some instances, and decreased heat transfer performance in other instances [15, 16]. In the case of an increased heat transfer performance, it was explained by an increased pressure difference and thereby an improved mass flow through the vanes [16].

Some manufacturers also produce brake discs with holes or grooves on the friction surfaces. The introduction of holes and grooves allows entrapped bulk flow, dirt and water, between the pad and the disc to escape, as well as that it increases the cooling area. As a result, an improved cooling performance and thereby a reduced risk of brake fading and pad glazing can be achieved. Studies have also shown that the introduction of holes can result in an increased mass flow through the disc and hence, an increased heat transfer [8]. However, the introduction of holes results in increased wear and stress concentrations, which, in turn can result in cracks and additional noise [17].

2.2 Governing Equations

The governing equations for a compressible, Newtonian fluid are the continuity, momentum and energy equation. Any CFD simulation involves solving the three equations which represent balance of mass, momentum and heat in a system. The system is said to be balanced when the sum of net transport and the net generation within the system, equals the sum of net accumulation within the system. The complexity of a simulation model is modified by adding/removing terms to/from these set of equations. The parameters of interest are then calculated by solving these equations.

2.2.1 Continuity Equation

The continuity equation, as given in equation 2.1, represents mass balance in a system [18].

$$\frac{\partial \rho}{\partial t} = -(\nabla \cdot \rho U) \quad (2.1)$$

The term in the right hand side represents the change of mass flux over time.

2.2.2 Momentum Equation

The momentum equation can be defined as the rate of momentum accumulation, equal to the sum of the rate of momentum into the system, the rate of momentum leaving the system and sum of forces acting on the system [18].

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_j} + \nu \frac{\partial}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + F_b \quad (2.2)$$

The terms in the left hand side represent inertial/acceleration terms, where the first term represents the time changes of the velocity in the studied point and the second term represents the influence from the surrounding flow on the velocity vector at the studied point. The right hand side of the equation represents force terms, where the first term is associated with the pressure field, the second field with the viscosity and the last term with the body forces.

2.2.3 Energy Equation

The various forms of energy that exist in a flow are total energy, kinetic energy, potential energy, chemical energy and thermal energy. Like momentum and continuity, for a system to be stable, there must exist an energy balance. The equation to model energy balance is given in equation 2.3 [18].

$$\frac{\partial h}{\partial t} = -\frac{\partial}{\partial x_j} \left[h U_j - \kappa \frac{\partial T}{\partial x_i} + \sum_n m_n h_n j_n - \tau_{kj} U_k \right] + S_h \quad (2.3)$$

The terms in the above equation from the left represent local energy change with respect to time, convective energy, $\kappa \frac{\partial T}{\partial x_i}$ is the heat flux due to conduction, $j_n = -D_n \frac{\partial C_n}{\partial x_j}$ is the diffusional heat flux of chemical species, $\tau_{kj} U_k$ is the irreversible transfer of mechanical energy into heat and S_h is the source term,.

2.3 Turbulent Flow

Turbulence is omnipresent in our daily lives. It is one of the most commonly occurring flow phenomenon and is one of the key elements in CFD. Turbulence significantly affects mass, momentum and heat transfer rates. It is a decaying process where large disturbances in the flow are formed by absorbing energy from the bulk flow which break down to smaller disturbances and eventually become laminar in nature. Turbulent flows can be characterized by being diffusive, irregular, chaotic and consist of a wide range of length scales, velocity scales and timescales.

Solving turbulent flow is very time-consuming, hence it is commonly modelled in the form of mathematical equations which can model the disturbances and its effect on the flow. Sometimes the information near the wall regions cannot be captured and needs extra steps in modelling, especially in the boundary layer region. The turbulent boundary layer can be divided into inner region and outer region based on the thickness from the wall. As shown in Figure 2.4, the inner region of the boundary layer can further be divided into viscous layer, buffer sub-layer and fully turbulent layer. The level upto which the flow properties have to be resolved is decided by the user and influences how the geometry is discretized.

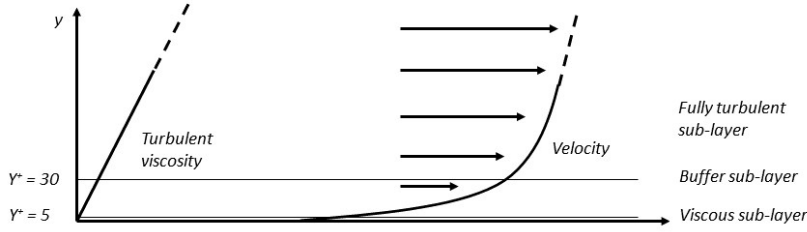


Figure 2.4: Sub-layers in the turbulent boundary layer [18]

Discretizing a geometry means dividing the geometry into smaller elements/cells of a known shape. In order to capture the flow properties in the boundary layer, care has to be taken during discretization. Each sub-layer in the boundary layer has an associated dimensionless distance from the wall, i.e., $y+$ value that helps in determining the first cell height. The user can decide upto which sublayer the flow properties have to be captured. This lets the user know the associated $y+$ value of the sublayer and in turn the first cell height. The $y+$ value is a function of the velocity of the flow in the first cell, u_* , the kinematic viscosity ν , and the distance to the wall, y as given in equation 2.4 and equation 2.5.

$$u_* = \sqrt{\frac{\tau_w}{\rho}} \quad (2.4)$$

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

The choice of the first cell height is dependent on the requirements of the turbulence model and wall function that are being used. Wall functions are semi-empirical rules that are based on the logarithmic law of the wall [18]. These functions help in avoiding dense meshes near the wall and are needed when a particular turbulence model is not valid in the viscosity affected region near the wall. They are used to estimate properties in the first cell close to the wall and hence demand a wise choice of first cell height near the wall. Taking into account the level of detail needed and computational time, a choice of wall treatment can be made between low $y+$, high $y+$ or an all $y+$ wall treatment, each having a different requirement of the first cell height. For a varying geometry with difficulty in achieving a consistent $y+$, an all $y+$ treatment can be used.

Some turbulence models solve one transport equation while others use two equations. In this work, a two-equation turbulence model will be used, which are discussed in more detail, in the following section.

2.3.1 RANS

Reynolds Averaged Navier Stokes equation (RANS) is based on statistical averaging of Navier Stokes (NS) equation. In RANS equation, each term of NS equation is decomposed to mean and fluctuating components, as can be represented by equation 2.6.

$$\phi = \phi' + \bar{\phi} \quad (2.6)$$

Where, ϕ can be e.g. velocity component, pressure, energy or species concentration. Since NS equation is non-linear, a non-linear acceleration term appears when averaging NS equation. This term is known as Reynolds Stress and cannot be solved directly, which is why it has to be modelled. The turbulence models for RANS provide the terms that are required to calculate the eddy viscosity ν_T , which is then used in Boussinesq approximation, equation 2.7, to model the Reynolds stresses. Two commonly applied turbulence models to do so, are the Realizable Two-Layer $k - \varepsilon$ and SST Menter $k - \omega$, which will be introduced in the following two subsections.

Realizable Two-Layer $k - \varepsilon$

$k - \varepsilon$ is a model to close RANS by introducing transport equations for the turbulent kinetic energy, k , and dissipation rate of k , ε , to obtain the eddy viscosity. In the standard $k - \varepsilon$ model, the normal stress can become negative for flows with large mean strain rates. In a realizable model, a correction factor that is a function of the local state of the flow, C_μ , is introduced in the turbulent kinetic energy equation. This is done by introducing Boussinesq approximation, equation 2.7, to analyze the normal components of the Reynolds stress and ensures that the normal stresses are positive for all flow conditions, making it realizable.

$$\langle u_i u_j \rangle = \Sigma_i \langle u_i^2 \rangle = \frac{2}{3} k - 2\nu_T \frac{\delta \langle U_i \rangle}{\delta x_j} \quad (2.7)$$

Another difference is that ε equation is modified in realizable $k - \varepsilon$ model, that makes the model not only predict planar flows, but also axi-symmetric flows. This also makes the realizable $k - \varepsilon$ model suitable for flows involving rotation and separation. The advantage of using a two layer model is that the entire boundary layer is resolved until the viscous sub-layer.

SST Menter $k - \omega$

In the $k - \omega$ turbulence model, transport equations are introduced for k and ω , to solve eddy viscosity, and thereby close RANS. The specific dissipation, ω , is used as the length-determining quantity, where $\omega \propto \frac{\varepsilon}{k}$.

The use of Shear Stress Turbulence (SST) Menter $k - \omega$ model, combines the best of two models i.e, it uses both $k - \varepsilon$ and $k - \omega$. The low Re turbulence near the wall is modelled using the $k - \omega$ model and is switched to $k - \varepsilon$ in the free-stream, making the SST model less sensitive to free-stream turbulence properties, as is a problem when using $k - \omega$ model alone. This property makes it usable in flows involving adverse pressure gradients and separating flow, although there is a risk of erroneous predictions in stagnation zones and regions with strong acceleration.

The use of this model has been proved advantageous, especially in the near wall regions as it eliminates the need for wall functions in the viscous sub-layer. However at low Re, there is a requirement of a very fine mesh close to the wall.

2.3.2 URANS

Unsteady Reynolds Averaged Navier Stokes (URANS) can be used when a long-term periodical oscillation is to be investigated. Turbulence fluctuations of flow quantities are not resolved in URANS approach. In this model, turbulence stress is introduced in the Navier-Stokes equation and have to be modelled using the available two equation models [19].

2.3.3 IDDES

The IDDES turbulence model utilizes a hybrid RANS-LES approach by using two equation model to model near wall turbulence and Large-Eddy-Simulation (LES) to model far field eddies. In this approach, RANS can be used in a much thinner near-wall region, in which the wall distance is much smaller than the boundary-layer thickness. Due to the fact that it utilizes LES, the required computational time will increase, but the accuracy and realizability will also increase due to it capturing turbulence better [20].

2.4 Heat Transfer

Heat can be transferred from the brakes through three modes; convection, conduction and radiation, which will be presented in the following subsections.

2.4.1 Convection

Convection is the main heat transfer mode in brake disc cooling when the disc is rotating. It is the heat transfer due to fluid motion and is defined by Newton's Law of Cooling, as shown in equation 2.8.

$$q = H\Delta T \quad (2.8)$$

Where, H is the heat transfer coefficient in W/m^2K , as will be introduced more into detail in the next section, ΔT is the temperature difference in K , and q is the convective heat flux in W/m^2 . Convection mainly comes from the internal surfaces, i.e. the vanes under the condition that the brake disc is spinning [21]. Since convection is dependent on cooling area and the temperature gradient, a higher cooling area or a higher temperature gradient will increase the convection heat transfer.

Convection can be categorized into two types, forced convection and natural convection. Convection due to rotation of a brake disc, a fan or a pump, i.e. convection due to external influence such as centrifugal forces is what is known as forced convection. On the contrary, if no external influence is present and the convection instead is due buoyancy forces, generated by temperature gradients resulting in density differences, the convection is called natural convection [22]. Brake discs are affected by both types of convection. Only considering convection, forced convection is dominant when a brake disc is spinning and natural convection is dominant when a hot disc is standing still. The ratio between the natural and forced convection can be defined by a coefficient, C , which relates the Rayleigh number (Ra), Prandtl number (Pa) and Reynold's number (Re), as shown in equation 2.9.

$$C = \frac{Ra}{PaRe^2} \quad (2.9)$$

When this ratio is equal or close to 1, mixed convection occurs, and if the ratio is bigger than 1 or less than 1, natural or forced convection dominates respectively.

2.4.2 Conduction

The second heat transfer mode is conduction. Conduction describes heat transferred through solids and air due to temperature gradients between regions in contact. Conduction can be explained by Fourier's Law as shown in equation 2.10.

$$q = \frac{\kappa\Delta T}{L} \quad (2.10)$$

Where, κ is the conductivity of the material W/K , ΔT is the temperature difference between the two regions in contact, L is the material thickness in m and q is the conduction heat flux in W/m^2 . Conduction is an important heat transfer mode since this mode transfers energy to the brake fluid, and can thereby result in brake fluid boiling. Generally, more heat is dissipated from the disc through conduction compared to radiation, but convection is still the main heat transfer mode for cooling of brakes [7]. The ratio between convective and conductive heat transfer is known as the Nusselt number, see equation 2.11.

$$Nu = \frac{q_{convective}}{q_{conductive}} = \frac{HL}{\kappa} \quad (2.11)$$

For a constant velocity cooling of a 500° C hot braking system, simulations have shown the Nusselt number to be 500 (inner friction surface) to 900 (outer friction surface), i.e. a much higher convective heat transfer compared to conductive heat transfer [7].

2.4.3 Radiation

Heat radiation is when heat is transferred from a heat source through electromagnetic radiation, i.e. through waves. Radiation has a temperature dependency, that can be derived from Stefan-Boltzmann law, as shown in equation 2.12.

$$q = \epsilon \sigma (T_1^4 - T_2^4) \quad (2.12)$$

Where, $\epsilon [-]$ is the emissivity of the surface, σ is the Stefan-Boltzmann constant in $W/m^2 K^4$, T_1 and T_2 are the temperature of the object and the surrounding medium respectively and q is the radiation heat flux in W/m^2 . Thus, when the temperature of the brake disc increases, the role of radiation becomes increasingly important. As the temperature drops, the effect of radiation decreases quickly since the radiation heat transfer is to the power of four dependent on the temperature. It has also been shown that most of the radiation of brake discs comes from the external surfaces [21]. However, in the context of brake disc cooling at lower temperatures, radiation plays a small role compared to convection and can usually be neglected [4].

2.5 Heat Transfer Coefficients

A commonly applied measure when studying convective heat transfer, is the convective Heat Transfer Coefficient (HTC). It describes the heat transfer occurring between a moving fluid and solid and can be defined, based on Newton's law of cooling, equation 2.8, according to equation 2.13.

$$H = \frac{q_s}{T_s - T_{ref}} \quad (2.13)$$

Here q_s is the local surface heat flux, T_s is the surface temperature and T_{ref} is the reference temperature. In order to estimate a suitable HTC, the reference temperature can be chosen in different ways. Commonly, the reference temperature is specified based on a user input, the first cell temperature or the temperature at a specified $y+$ value. For hot brake discs, the average HTC has been found to vary in the range from 25-60 $W/m^2 K$ [6, 8, 23]. For normal operating conditions the HTC is found to be higher on the vanes compared to the friction surfaces.

In Star-CCM+, local flow conditions, commonly estimated from standard wall functions, are used to model the convective heat transfer at the walls for turbulent flows according to equation 2.14.

$$q_s = \frac{\rho_f(y_c) C_{p,f}(y_c) u_*}{T^+(y^+(y_c))} (T_s - T_c) \quad (2.14)$$

Where, f is subscript for fluid and c is subscript for near wall cell, meaning that ρ_f is the fluid density, $C_{p,f}$ is the fluid specific heat, u_* is the reference velocity and T^+ is the dimensionless temperature, all in the near wall cell. T_s and T_c are the surface temperature and near wall cell temperature respectively. For Star-CCM+ to solve the HTC, equation 2.14 is used to solve for the convective heat flux based on local flow conditions, followed by solving the HTC from equation 2.13. Star-CCM+ has four built in functions for post-processing of HTCs, which based on the chosen reference temperature might yield different results, as will be explained in the following subsections.

2.5.1 Specified Temperature Heat Transfer Coefficient

The field function known as the "Heat Transfer Coefficient" in Star-CCM+, which in this report is denoted as Specified Temperature HTC, uses a user specified reference temperature, $T_{ref} = T_{user}$, to evaluate the HTC according to equation 2.13. Guidelines state that the inlet or bulk temperature should be selected for internal flows and the free stream or environmental temperature should be selected for external flows.

The advantages of using this definition of the reference temperature is that the reference temperature is not first cell height nor velocity dependent. However, when the temperature is changing throughout the domain, the HTC might not yield correct predictions [24].

2.5.2 Local Heat Transfer Coefficient

The "Local Heat Transfer Coefficient" field function in Star-CCM+ uses the temperature in the cell closest to the wall, i.e. $T_{ref} = T_c$, to estimate the HTC. This means that by rewriting equation 2.13 as a function of convective heat transfer, implementing $T_{ref} = T_c$ and equating it to equation 2.14, the HTC can be written based on local flow conditions only, as seen in equation 2.15.

$$H = \frac{\rho_f(y_c)C_{p,f}(y_c)u_*}{T^+(y^+(y_c))} \quad (2.15)$$

Thereby, the user does not have to specify any reference temperature. However, the function uses the near wall cell temperature, meaning that it will be first cell height dependent. This means, that if the first cell has a low cell height, i.e. a low y^+ value, the value of reference temperature will be approximately the same as the wall temperature, possibly resulting in high values of the HTC. On the contrary, for a high y^+ value, the reference temperature will be similar to the flow temperature, thereby predicting lower HTC values.

The local HTC is applicable when y^+ is within the range of $30 < y^+ < 150$. If $y^+ > 30$ cannot be achieved, e.g. in a narrow path between two walls or in the vanes of a brake disc, the measure will predict high HTC and possible erroneous values, as formerly presented [24].

2.5.3 Specified y^+ Heat Transfer Coefficient

A third function to quantify the heat transfer in Star-CCM+ is the "Specified y^+ Heat Transfer Coefficient". This field function requires the user to specify a y^+ value at which the reference temperature will be computed, i.e. $T_{ref} = T_{y^+_{ud}}$.

Due to the fact that the reference temperature is computed at a set y^+ value, the reference temperature will not be first cell height dependent and thus, the Specified y^+ HTC is not first cell height dependent. One downside is that it might be hard to compare cases with different velocities, since an altered velocity will affect the shear stresses. In turn this will affect the first cell velocity and thereby the y^+ value according to equation 2.4 and equation 2.5. This would result in using different reference temperatures for the comparison and thus, compromise the comparison.

The Specified y^+ HTC is applicable for studying flow problems with a turbulent regime where temperature might be varying. It is recommended that the y^+ is in the range of $30 < y^+ < 150$. The recommended y^+ value to evaluate the reference temperature is $y^+ = 100$ [24].

2.5.4 Virtual Local Heat Transfer Coefficient

The final field function for investigating the HTC in Star-CCM+ is the "Virtual Local Heat Transfer Coefficient". It is similar to the Local HTC and is first cell height dependent due to it utilizing equation 2.15. However, this field function can be used without solving the energy equation, meaning that simulations require a shorter run-time to estimate the Virtual Local HTC compared to the simulations where the previously explained HTC field functions are estimated. In order to solve the Virtual Local HTC, the user has to specify the Prandtl number, turbulent Prandtl number and the specific heat. The Prandtl number is defined as the ratio of momentum to thermal diffusivity and approximately varies between 0.7-0.72 for air in the range of $0 - 600^\circ\text{C}$ and 1 atm. The turbulent Prandtl number is similar to the Prandtl number, but instead relates the momentum eddy to thermal eddy diffusivity. It is commonly specified to be 0.9, since this is what is used if the energy equation is solved based on a Boussinesq approximation. Star-CCM+ needs these parameters for the standard wall functions to be able to solve the dimensionless temperature for equation 2.15, since these are not available when the energy equation is not solved. For the standard wall functions to be accurate, thereby also the reference temperature, it is recommended that the y^+ value is in the range $30 < y^+ < 150$ [24].

2.6 Discretization

Meshing is a process of discretizing the geometry into smaller elements of known shape, i.e. cells. A good quality mesh plays an important role as it directly affects the rate of convergence, accuracy and run-time of the simulation.

Broadly, meshes can be classified as structured, unstructured and hybrid meshes. In a structured mesh, the elements are connected in a regular fashion while in an unstructured mesh, the elements are connected in an irregular fashion as displayed in Figure 2.5.

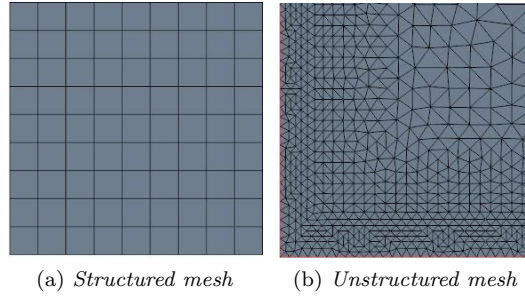


Figure 2.5: *Mesh Types.*

The advantage of a structured mesh is that it may result in easier and quicker convergence. However, the time required to set up a structured mesh is usually higher. Applying a structured mesh might also introduce additional problems with mesh quality and over-simplification of a geometry since it might have a hard time capturing details unless the mesh is very fine. On the contrary, an unstructured mesh is less time-consuming to set-up, generally results in less elements but may result in more difficulty in achieving convergence [25]. A third type of mesh is what is known as a hybrid mesh. A Hybrid mesh has a mix of structured and unstructured meshes in different regions, thereby making it possible to utilize the advantages of both mesh types throughout the domain.

The mesh may consist of different elements with different advantages and disadvantages. Typically, a 3D mesh may contain tetrahedron, triangular prism, hexahedron, polyhedron and pyramid cells. Two element types commonly applied in unstructured mesh are tetrahedral cells and polyhedral cells, as shown in Figure 2.6.

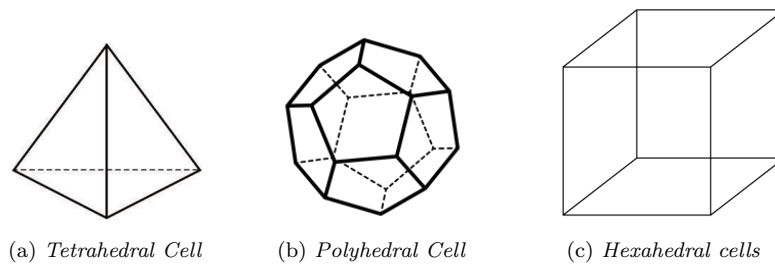


Figure 2.6: *Typical element types.*

Studies have shown that a polyhedral mesh is advantageous over a tetrahedral mesh, since it produces similarly accurate results and results in less elements and hence, faster convergence [26].

In structured mesh, hexahedral elements are commonly applied, also showcased in Figure 2.6. Using hexahedral cells to construct a mesh, the advantages of a structured mesh are gained. In Star-CCM+, a structured mesh with hexahedral elements can be achieved by applying the trimmed mesher. The trimmed mesher applies a hexahedral mesh and trims the elements on boundaries of the region. These elements then become hexahedral cells with edges or corners cut off, i.e. polyhedral cells. However, most of the grid will still consist of hexahedral

cells, whereby a high quality grid is produced that can handle complex geometries [24].

Near the object of interest, gradients commonly become greater than in the bulk flow. To capture this, the element size has to be finer in regions where high gradients are expected, e.g. at the walls of an object. If the element size is not refined, the gradients might not be properly captured, whereby erroneous results can be achieved. To avoid this, the cells near the wall are commonly refined using what is known as prismatic cells. Prismatic cells are orthogonal cells produced next to the wall to improve the accuracy of the solution and better capture the gradients.

2.6.1 Mesh Independent Solution

One criteria for simulations to well resemble reality is that the mesh has to be fine enough to capture and estimate reality. If a too coarse mesh is applied, the gradients might be too big for the mesh to capture, thereby generating erroneous results. To avoid this, a mesh independent solution should be found, i.e. the solution should not change significantly for an increase in number of cells.

2.7 Modeling of Rotation

Simulation of a rotating object can be done using several different approaches, such as Moving Wall (MW), Moving Reference Frame (MRF) or Rigid Body Motion (RBM). MW and MRF are steady-state approaches whilst RBM is a unsteady approach. These methods have different advantages and drawbacks which will be presented in the following section.

2.7.1 Moving Wall

The MW method is a steady state approach in which the mesh does not move. To implement the technique, a tangential velocity is applied to the surfaces that are supposed to rotate. The fluid around the object is not given a rotation condition as is the case for MRF and RBM. These conditions result in that the disc boundaries will be simulated as rotating, but with a low interaction with the flow field.

For ventilated brake discs, it is important to capture the air pumped through the vanes, i.e. the radial flow. Due to the low interaction between the disc and flow field when using the MW approach, this technique has a hard time capturing the radial flow. However, compared to a unsteady approach such as Rigid Body Motion, the computational cost is significantly lower [24].

2.7.2 Moving Reference Frame

To apply a Moving Reference Frame, a reference frame is created and applied to the moving region. In the moving region, a constant grid flux of the volume flow rate is introduced into the conservative equations, which is calculated based on the local reference frame instead of the global reference frame. Additionally, forces imposed by rotation are introduced. These implementations make it possible to run the simulation in steady state by replicating the effect of constant rotating motion instead of moving the cells as is the case for RBM.

The guidelines for the moving reference frame method state that the method should not be applied if there is a noticeable component of flow perpendicular to the axis of rotation. The reason being that the method cannot capture flow perpendicular to the axis of rotation and is only accurate for flow close to the rotating object that is approximately symmetric around the axis of rotation. If there is high interaction between the object and flow field the previously mentioned conditions might not be met, whereby MRF might produce inaccurate results, especially close to the object.

The advantage of this approach is that the required run-time is low compared to RBM. However, when applying MRF there is a compromise in the accuracy and realizability since it can result in unrealistic result at a detailed level. Consequently, the method is good for overall predictions, but may not be the best choice in

detailed studies [24]. Previous studies have shown that for an overall prediction of the HTC, experiments and simulations using MRF correlated within a few percents [3].

2.7.3 Rigid Body Motion

A third rotation technique, which is not simulated in steady state, is RBM. The reason for RBM to be a unsteady approach is due to the fact that it moves cell vertices around a specified axis in time. Similar to MRF, RBM introduces a grid flux of the volume flow rate into the conservative equation and body forces imposing rotation.

The advantage with a unsteady approach, such as Rigid Body Motion, is that in most applications it will result in a higher accuracy and higher realizability since it does not assume the solution to be steady. Thereby, it can better capture the interaction between the object and the flow interacting with it [27]. However, this comes at the price of a higher computational cost due to the unsteady nature of the simulation [24].

2.8 Field Functions

To explain and understand the flow field and heat transfer from brake disc, post-processing of the results is required. Star-CCM+ includes different ways to post-process, e.g. by monitoring parameters and investigating contours and vectors. Two frequently applied parameters, known as field functions in Star-CCM+, which will be explained below are the pressure coefficient and the skin friction coefficient.

2.8.1 Pressure Coefficient

When post-processing simulations, static pressure is regularly a parameter of interest since it can easily be interpreted and can be used to explain flow phenomena occurring. According to Bernoulli's principle, a high pressure area characterizes an area of low velocity and vice versa.

Due to the fact that absolute pressure is usually dependent on the velocity magnitude, it can be hard to compare pressure distributions for simulations with different velocities. However, this can be overcome using the coefficient of pressure, C_p , instead of the absolute pressure. The coefficient of pressure, equation 2.16 describes the pressure in relative measures by normalizing the pressure with the free stream velocity square and density, making it easier to compare simulations with different velocities.

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2} \quad (2.16)$$

Here p is the static pressure at the point of evaluation of the coefficient of pressure, and p_∞ , ρ_∞ and U_∞ are the pressure, density and velocity in the free stream flow.

2.8.2 Skin Friction Coefficient

Similar to pressure, wall shear stress is commonly used in flow field investigations, since it can be used to characterize if the flow is attached or separated. As the shear stress, equation 2.17 is equal to zero, the flow is detached.

$$\tau_w = \mu \left(\frac{du}{dy} \right)_{y=0} \quad (2.17)$$

Here μ is the dynamic viscosity, u is flow velocity parallel to the boundary and y is the height above the wall, which is equal to 0 as the wall shear stress is evaluated. As is the case with pressure, the wall shear stress is dependent on the flow velocity, making it hard to compare cases with varying velocities. To overcome this problem, the skin friction coefficient, equation 2.18, is commonly applied.

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho_\infty U_\infty^2} \quad (2.18)$$

Here the wall shear stress is normalized with the density and the free stream velocity.

3 Methodology

In the following chapter the applied methodology and employed settings are presented through three sections; Pre-processing, Simulation Set-up and Post-processing. The Simulation Set-up section is divided into the settings for the baseline configuration, which is explained in the 'mesh' and 'solver settings' section, followed by the 'investigations performed' section which presents the altered settings for the various investigations.

3.1 Pre-processing

To be able to run and perform post-processing of the simulation results, it is first required to pre-process the received mesh files and geometry files. The files received consisted of either a surface mesh or a geometry part file with only one surface. To set up the simulation in Star-CCM+, different surfaces have to be defined in the geometry file, in order for the boundary conditions to be applicable in Star-CCM+. This is accomplished using the pre-processing tool ANSA. In ANSA the built in functions for filling holes with meshed elements and defining new PID's, make it possible to generate five different surfaces: outlet, inlet, inner friction surface, outer friction surface and skin, as seen in Figure 3.1.

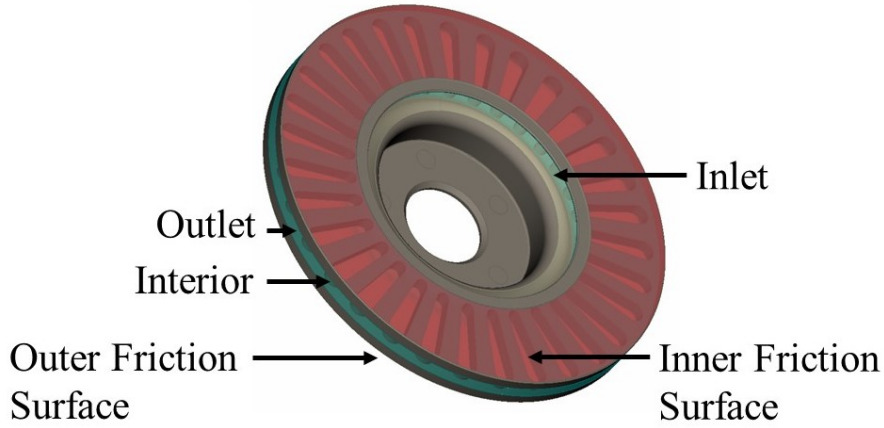


Figure 3.1: *Surfaces defined in Ansa.*

To simulate the rotation of the brake disc with MRF or RBM, explained in Section 2.7, an air region for the vanes has to be set-up. The reason being, that the rotating air region will be simulated in a separate reference frame and thereby cannot be the same region as the disc. To do so, the surface mesh has to be divided into two different regions, after which a new surface can be generated on top of the new regions, in order to close the split regions. This will result in two regions as displayed in Figure 3.2.

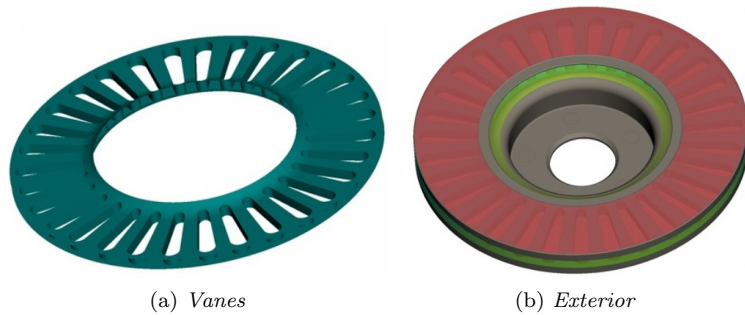


Figure 3.2: *Vane and exterior geometry.*

When starting from a geometry file, the procedure is similar to starting from a surface mesh file, except that a surface mesh has to be generated, not only on the inlet and outlet, but also on the geometry. To increase the resolution of the mesh, the spacing of points is defined with a target cell length of 0.5 mm. The resulting mesh files can then be exported in stereolithography (.STL) format from ANSA to be imported into Star-CCM+.

3.2 Simulation Set-up

This section will present the settings and values applied to generate the mesh and run the simulations in Star-CCM+. First, the setting for the baseline configuration used for steady-state investigations will be presented. Subsequently, with the baseline defined, the settings applied in the studies performed to answer the research questions will be explained.

3.2.1 Discretization

After pre-processing in ANSA, the .stl files can be imported as a surface mesh files into Star-CCM+. To place the disc geometry into a fluid domain in the simulation, a 2x2x2 meter cube is created. The imported parts can then be placed at $x = 0.7, y = 1, z = 1$ meters. Consequently, the domain is split into two regions using the subtract operation; the vane region that will have different boundary conditions when applying moving reference frame and the domain, which includes the surface of the disc and the walls of the domain.

Since the mesh and wrap applied in ANSA were only generated to import a high quality geometry into Star-CCM+, new mesh operations were carried out on the imported parts. Due to the nature of geometry of the vanes, generating a structured mesh required additional effort. Therefore, and that it has shown to perform well for rotation and heat transfer as explained in Section 2.6, an unstructured mesh was generated. The polyhedral mesher is applied in combination with the prism layer and surface remesher. For the domain and the brake disc, the trimmed cell, prism layer and surface remesher operations are applied to gain the advantages of a structured hexahedral-dominant mesh, as explained in 2.6. Similar to generating a mesh for the vanes and the domain, a solid mesh is generated for the disc. The element size is chosen to be big, since the gradients are not expected to be significant in the solid disc when compared to the domain. Due to the shape of the disc and that it will transfer heat, a polyhedral mesher and surface remesher are applied. The mesh parameters applied for the mesh operations are presented in Table 3.1.

Table 3.1: Mesh parameters applied.

Settings	Domain	Vanes	Disc
Base Cell Size [mm]	50	4	18
Target Surface Size [mm]	25	1.2	9
Minimum Surface Size [mm]	5	0.4	1.8
Number of Prism Layers	2	1	-
Prism Layer Stretching	1.3	1.3	-
Prism Layer Total Thickness [mm]	4	2.5	-

To avoid generation of prism layers on the domain walls and interfaces, the boundary conditions and interfaces, presented in the following section, have to be specified before the executing the mesh operations.

In addition to this several refinements are added where gradients are expected to be high, i.e. in the wake region and around the brake disc, as can be seen in Figure 3.3.

These refinements control the cell size of the trimmed cells by setting the isotropic cell size to 50 % of the base size (reference cell size) in the outer refinement and 20 % of the base size in the inner refinement. For the wake refinement, a wake of 250 mm with 3 mm trimmed cells is generated. These settings resulted in a mesh with 1.8 million cells.

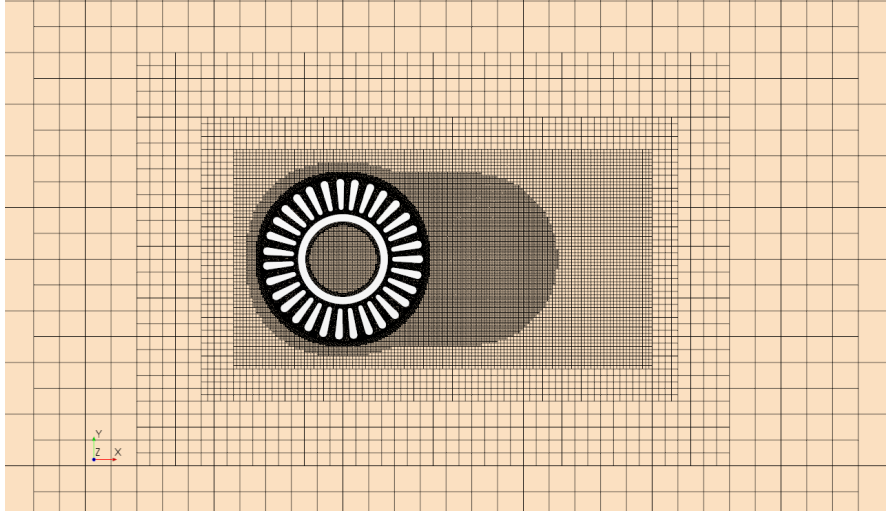


Figure 3.3: *XY-plane visualizing the mesh and applied refinement regions. Note that the solid mesh is not displayed in the figure.*

3.2.2 Solver Settings

As mentioned in the previous section, the boundary conditions should be specified before meshing. Since the baseline configuration utilizes a MRF set-up to mimic rotation of the vanes, a rotating reference frame spinning at 800 rpm, equaling about 100kph, has to be created. To specify the vane region to rotate, the rotating reference can be specified as the motion specification of the vane region. For the brake disc surfaces, MW conditions, i.e. a tangential velocity specification of 800 rpm, have to be applied to the disc surfaces. In order to simulate a heated brake disc, static temperature boundary conditions of 520 °C are applied to the vane and disc boundaries due to it being close to the expected temperature of braking the brake disc at 100 kph [1]. The domain boundaries are specified as symmetry walls, a velocity inlet with a temperature of 20 °C and an inlet velocity of 3 m/s to simulate oncoming flow from inside the wheel-house, and a pressure outlet.

Having defined the boundary conditions and generated a suitable mesh, the models for solving the simulation have to be defined. The following models are applied to take convective heat transfer and the turbulent flow field in to consideration in a steady state simulation,

- Coupled energy
- Coupled flow
- Exact wall distance
- Gas
- Gradients
- Ideal Gas, Incompressible
- K-epsilon Turbulence; Realizable k-epsilon Two-layers
- Reynolds-Averaged Navier-Stokes
- Steady
- Three dimensional
- Turbulence
- Two-Layer All y+ Wall treatment

For the solid region, i.e. the brake disc, another continua is specified according to the following properties,

- Constant density
- Coupled solid energy
- Gradients
- Solid
- Steady

- Three dimensional

For the solver, a 2nd-order implicit solver is specified since it was stable enough to achieve convergence. To achieve a better initialization than the standard option provides, the grid sequencing expert initialization solver was also applied. To avoid divergence but still achieve quick convergence, a linear ramp is selected for the courant number, ramping the courant number from 0.1 to 20 over the initial 100 iterations.

3.2.3 Convergence

Having set-up the simulation, convergence is determined by investigating monitors of drag coefficient, mass flow through the disc and heat transfer from the disc. When these quantities stabilize and the residuals of continuity, momentum, energy, turbulent dissipation rate and turbulent kinetic energy dropped by 10^3 , the simulations were judged to be converged. For the unsteady simulations, explained in the following section, three additional points (probes) monitoring velocity are set up. The unsteady simulations are judged to be converged when the inner iterations converge as well as that a continuous pattern for the velocity monitors can be observed.

3.2.4 Investigations

As explained in the introduction, Section 1, this thesis aims at answering how well Star-CCM+ can be used to capture brake disc cooling performance. In the process of finding out this, several investigations were performed to answer the research questions stated in Section 1.4. This section will cover the altered settings for each of these studies.

Baseline configuration and mesh dependency study

Firstly, investigations on how to set-up a baseline configuration were carried out as per the methodology explained in section 3.2. To find out how many elements should be used for the mesh, a mesh independency study was performed for eight different cases, where the target cell size was varied as mentioned in Table 3.2.

Table 3.2: Cell size specifications for the meshes used in the mesh dependency investigation.

Case	Number of million cells	Target cell size domain [mm]	Target cell size interior [mm]	Minimum cell size domain [mm]	Minimum cell size interior [mm]
1	0.08	200	12	40	4
2	0.10	150	9	30	2.4
3	0.20	100	4.8	20	1.6
4	0.44	55	2.64	11	0.88
5	0.86	37.5	1.8	7.5	0.6
6	1.8	25	1.2	5	0.4
7	2.8	19	0.96	4	0.32
8	3.6	16	0.77	3.2	0.26

After running these simulations until convergence, the mass flow through the disk and the coefficient of drag were plotted against the number of cells to see if a grid independent solution was achieved. To investigate grid independence, MRF was used since it is fairly quick compared to the RBM and significantly more accurate compared to moving wall. The turbulence model used was the Realizable k- ϵ two-layer. The grid independent solution was said to be achieved when the properties, in this case mass flow through the disk and coefficient of drag, stopped fluctuating.

For the baseline configuration, the rotation techniques; MW, MRF and RBM were also investigated with Realizable k- ϵ two-layer as the turbulence model. MW is set-up simply by specifying a tangential velocity for the boundaries that should simulate rotation. For MRF, a reference frame has to be set-up as explained in the previous section. For the case simulating RBM, the simulation has to be unsteady, i.e., the continua was changed from steady to implicit unsteady, a rotation motion had to be defined under motions, a local rotation

rate boundary condition should be specified as well as that the time-step should be defined as 1 degree per outer iterations. The inner iterations were set to 20. Since a unsteady simulation will yield different results at different times in the simulation, mean monitors of quantities of interest have to be defined for post-processing purposes. When an converged solution was achieved, the mean monitors were reset in order to only record one full rotation to post-process.

Additionally, the Realizable $k-\varepsilon$ two-layer and SST $k-\omega$ turbulence models were investigated to see which turbulence model is the most suitable for the steady state baseline configuration. For these investigation models, MRF was used as the rotation technique. For the unsteady simulations, IDDES and URANS ($k-\varepsilon$) were compared with RBM as the rotation technique.

To determine which rotation technique and turbulence model performed the best, the computational time, mass flow through the disc, heat flux and contours of velocity and coefficient of pressure were investigated.

Heat Transfer Coefficients

Having defined the baseline configuration, the simulation set up was used to carry out investigations of HTC definitions. Since the HTC can be defined in different ways as explained in Section 2.5, the HTC's values were compared for three different meshes where the first cell height was altered according to Table 3.3.

Table 3.3: Specification of first cell height for first cell height dependent HTC.

Case Number	First Cell Height [mm]
Case 1	1.9
Case 2	2.7
Case 3	4.2

In addition to this, the performance of the Virtual Local HTC was investigated by comparing simulations with and without the energy equation solved, for a varying rotational velocity from 400 to 900 rpm. To determine how well the Virtual Local HTC captured and reflected the Specified Temperature HTC, the Virtual Local HTC and Specified Temperature HTC were normalized with their respective value at 800 rpm, in order to be able to compare them together. From this the percentage increase for both the Virtual Local HTC and the Specified Temperature HTC between certain rotational velocities could be compared. To further investigate the performance of the Virtual Local HTC, simulations with energy solved and not solved were also compared for three different brake disc geometries.

Lastly, the investigation of HTC's included an investigation to see if it is possible to linearize, extrapolate and scale the Specified Temperature HTC based on disc rotational velocity and temperature, since if possible, it could be used for quick estimations. To investigate linearity based on rotational velocity, the Specified Temperature HTC was plotted both for the vanes and friction surfaces as well as that a gradient based on the interval 400-900 rpm was estimated. This gradient was then used to extrapolate the Specified Temperature HTC values at 200 and 1600 rpm. To determine how well the prediction matched, simulations were then run to verify these values. For the linearity based on temperature, the gradient based on the slope between 420-520 °C was used and investigated against Specified Temperature HTC values in the range of 120-920°C. Finally, the possibility of scaling a non-linear temperature dependent HTC function with regards to rotational velocity was investigated. In order to do so, a function describing the HTC as a function of temperature was predicted based on the simulations run at 800 rpm. This curve was then scaled with the average deviation from three HTC points at 120, 520, 920 °C, for a rotational velocity of 400 and 1200 rpm respectively. To find the correlation, the points from the simulations were plotted and compared with the predicted non-linear HTC functions.

Parameter dependencies

With the most suitable HTC definition determined, the heat transfer dependencies on free-stream velocity, rotational velocity, ambient temperature and disc temperature were investigated to better understand the

cooling performance of the brake disc. This meant changing the boundary conditions as mentioned below:

- Ambient velocity: 0.1, 1, 3, 6, 9 m/s
- Disc rotational velocity: 600-900 rpm with 100 rpm intervals
- Ambient temperature: -10°C - +30°C with 10°C intervals
- Disc temperature: 320°C-620°C with 100°C intervals

Based on the simulations, the heat transfer for the vanes and friction surfaces as well as the mass flow was plotted against the parameter investigated.

Disc designs

To further understand how the brake disc geometry affects the cooling performance, three common vane configurations: radial, tangential and curved vanes were investigated. These were investigated applying the settings from the baseline configuration on different geometries. All discs investigated had a diameter of 320 mm, a thickness of 28 mm and 40 vanes. For the radial vane brake discs, the vane width was altered according to Table 3.4.

Table 3.4: Specification of vane thickness for radial vane designs investigated.

Case	Inner Vane Width [mm]	Outer Vane Width [mm]
Thin Vanes R1	3.7	8.3
Medium Vanes R2	7.7	13.7
Wide Vanes R3	10.7	17.8

In case of the tangential discs, the vane thickness was kept constant as the dimension of "Medium Vanes R2", as shown in Table 3.4. Instead, the vane tangent was initially altered from 50° to 60° in a step of 2.5°. More variations, from 0°, i.e. a radial disc, to 60°, as shown in Table 3.5 had to be studied, in order to find a trend and a peak performance point.

Table 3.5: Specification of vane tangent for tangential vane designs investigated.

Case No.	Vane Tan- gent [deg]
Case T1	0
Case T2	10
Case T3	20
Case T4	30
Case T5	40
Case T6	50
Case T7	60

Lastly, curved vane brake discs were also investigated. Similar to the tangential discs, the vane thickness was kept constant as per the specification of "Medium Vanes R2" in Table 3.4. Instead, the curvature of the vane was changed by altering the location of the inner and outer vane point according to Table 3.6

Rotation direction

So far the simulations carried out were in anti-clockwise direction and as discussed in the theory section, the tangential and curved vane type are direction dependent. The influence of changing the rotation direction was studied by rotating the tangential disc mentioned in table 3.5 in the clockwise direction for a rotational velocity of 800rpm and same settings as the baseline configuration.

Table 3.6: Specification of inner and outer vane point for curved vane designs investigated.

Case Number	Inner Vane point	Outer Vane point
Case C1	20	55
Case C2	22.5	57.5
Case C3	25	60
Case C4	27.5	62.5
Case C5	30	65

Additional parts

The influence of additional parts, such as brake pad-calliper combination and dust shield was also studied. For this study, a new geometry was provided with the shield, calliper and pads. The dimensions of the geometries were different from the other discs used in the previous studies. Pre-processing of the geometry was done in a similar fashion as mentioned in section 3.1. Due to the complex geometric features of the additional parts and small gaps present in the geometry, there are possibilities of generating an intersecting mesh. Hence, the additional parts were surface wrapped using a variable length surface wrapper with a minimum element size of 0.5mm and a maximum element size of 10mm in ANSA. The parts were then imported into Star-CCM+ and rotated and moved to the centre of the domain, i.e. $x = 0.7, y = 1, z = 1$. With the parts positioned, the added parts were subtracted from the domain, and the mesh operation defined in 3.2.1 was re-run. The simulations were carried using the baseline configuration settings and for a rotational velocity of 800rpm.

Gravity

In order to capture buoyancy effects, gravity should be modelled. Since gravity is expected to be of importance for natural convection cases, but not for forced convection cases, the effect of gravity for different rotational velocities, 0, 100, 400, 800, 1200 rpm was investigated. At 0 rpm and 100 rpm, the inlet velocity was set to 0 and 1 m/s respectively. For the remaining cases, the inlet velocity was set to 3 m/s . The temperature was set to 520 °C.

When the gravity model is included in the physics, three additional reference values have to be specified; “Gravity”, “Reference Altitude” and the “Reference Density”. In this study these values were set to $-9.81m/s^2$, $[1.0, 1.0, 1.0]m$ and $1.225kg/m^3$ respectively.

Additionally, all domain wall boundary conditions, except for the ground, were changed to pressure outlets. When forced convection is expected to dominate, the “Two-Layer Type” of the turbulence model should be set to “Shear Driven (Wolfstein)” whilst when natural convection is expected to dominate, i.e. for a rotational velocity of 0 rpm, the Two-Layer Type should be set to “Buoyancy Driven (Xu)”. For the natural convection case, the wake refinement was altered from behind the disc to on top of the disc to properly capture the buoyancy driven flow.

Radiation

The baseline configuration is set-up only to capture convective heat transfer. However, to better replicate reality radiation also has to be included. To study the effect of radiation, simulations including the models were compared to simulation without the models required to capture radiation for four different rotational velocities: 600, 700, 800 and 900 rpm, as well as that the radiation heat transfer values were compared to hand-calculated values.

To capture radiation in Star-CCM+, the following additional models have to be added to the physics.

- Radiation
- Surface-to-Surface Radiation

- Gray Thermal Radiation

The Surface-to-Surface radiation model is suitable since it captures thermal radiation between diffusive surfaces, which is what is expected from the disc to the domain. Due to the fact that the temperature is expected to change in a moderately small interval, the wavelength and frequency are not expected to change much, whereby the Gray Thermal Radiation models is suitable since it calculates the radiation independent of wavelength.

By including these models, an additional quantity, "Radiation Temperature" has to be specified in order to model the radiation properly. The radiation temperature is required since the model sets the emissivity to unity and models the thermal environment as a black body. In this case the radiation temperature is set to the ambient temperature, i.e. 20 °C. Furthermore, the emissivity has to be specified for the radiating surfaces. Depending on the condition of the brake disc, the emissivity can vary significantly. In this study, the emissivity was set to 0.5, which replicates a used but not worn out disc. The reflectivity was set to "Auto Calculate" and the transmissivity was set to 0.

Solid mesh

Similarly as for the baseline configuration, a mesh independency investigation was performed for the solid mesh. To save computational time, the simulations were not performed in unsteady, instead the simulation set-up was altered to simulate rotation and heat input from a pad in steady state.

To simulate the "unsteady heat input" in steady-state, two additional surfaces with the same dimensions as the friction surfaces, and two pad-like surface, both with a thickness of 3 mm were generated in ANSA, as displayed in Figure 3.4.

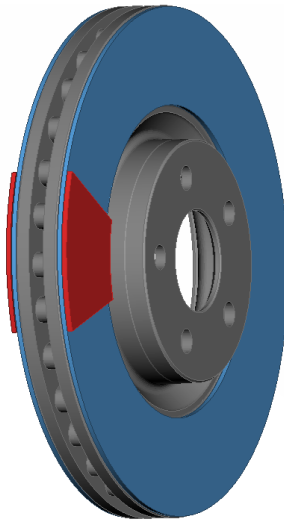


Figure 3.4: *Additional surfaces generated in ANSA to simulate heat input and rotation in steady-state.*

These were then imported into Star-CCM+ with a 1 mm gap between the surfaces. The pad surfaces were specified with a total energy source boundary condition and had the same continua as for the solid disc. For the heat from the pads to be distributed without the disc actually rotating, the extra surfaces on top of the friction surfaces were set with an additional continua similar to the one for the disc, except that it had an anisotropic thermal conductivity. In the region for these surfaces, the anisotropic thermal conductivity was specified as an axisymmetric tensor with a cylindrical coordinate system, an axial component of $10^6 W/mK$ and a cross-stream component of $24 W/mK$. The pad-like and extra surfaces were then added to the subtract region, meshed with similar settings to the brake disc case number 6, and interfaced against the domain, disc and themselves. The meshers applied were already explained in the baseline configuration, Section 3.2, and the settings used for the mesh independency investigation of the solid mesh for the brake disc are displayed in Table 3.7.

Table 3.7: Cell size specifications for the meshes used in the solid mesh dependency investigation.

Case	Number of thousand cells	Target cell size [mm]	Minimum cell size [mm]
1	7	24	9.6
2	18	15	6
3	52	12	2.4
4	75	9	1.8
5	145	6	1.2
6	327	3	0.6

Cool-down Simulations

To minimize the required computational run-time, whilst still capturing the cool-down over time, the unsteady cool-down simulations were run only with the solid region. Since the aerodynamic flow does not have to be solved, the time-step can be increased and was set to 1 second. The inner iterations were specified to 4. Having no fluid region present, means that there will be no fluid to cool the disc, whereby a convection boundary condition, with an ambient temperature and HTC, had to be specified to the disc boundaries.

In the investigation of the linearization of the HTC's, Section 4.2.4, it was found that the HTC is not entirely linear, which is why the functions, describing the HTC as a function of temperature, were used as the HTC boundary conditions. This made it possible to simulate a disc cool-down from 520 °C to 20 °C for a rotational velocity of 50, 100 and 150 kph.

3.3 Post-processing

Lastly, after having achieved converged solutions, the results could be post-processed. The post-processing was performed both studying parameters and absolute values, as well as studying contours of the flow field throughout the domain. The main parameters of interest for post-processing purposes were the Specified Temperature HTC, the massflow, the coefficient of drag, the heat transfer and the heat flux. This was combined with contours of the pressure coefficient, skin friction coefficient, velocity magnitude, Specified Temperature HTC and velocity vectors to understand and explain the flow field.

4 Results and Discussions

This chapter will follow a similar layout to the methodology section. Firstly, the results acquired to define the baseline configuration and the study of HTC's will be presented, in order to be able to use these for further post-processing purposes throughout the report. Next, the results from the parameter dependencies, disc designs and rotation directions, additional parts, gravity, radiation, solid mesh and unsteady cool-down simulations will be presented, combined with a discussion as the base to answer the research questions in Section 1.4.

4.1 Baseline Simulation Set-Up

This section will present the results attained for the baseline configuration and is divided into three subsections covering the results of: the mesh, rotation techniques and turbulence models.

4.1.1 Mesh Dependency Investigation

The mesh dependency investigation was carried out for eight different meshes, ranging from a total number of cells of 0.08 million cells to approximately 3.6 million cells. The specifications, such as the minimum cell size and target cell size are displayed in Table 3.2 in Section 3.2.4. To determine if the solution was mesh independent, the drag coefficient and mass flow through the disc combined with contours of coefficient of pressure and velocity magnitude were monitored. Figure 4.1 illustrates how the drag coefficient, mass flow and the computational time was affected by the increased number of elements in the mesh.

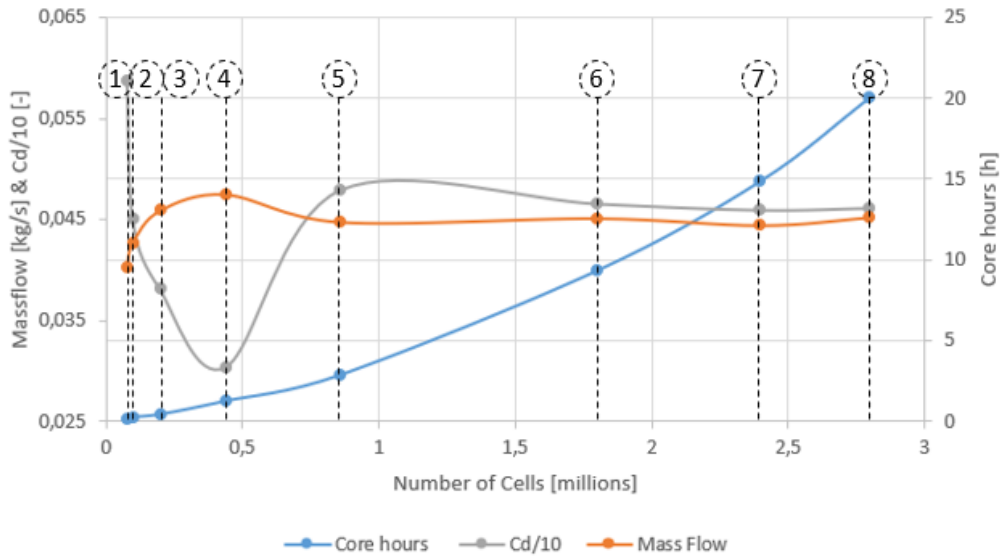


Figure 4.1: *The computational time, drag coefficient divided by ten and the mass flow average through the disc as a function of number of cells.*

Based on Figure 4.1, one can observe the stabilization of the drag coefficient and mass flow average through the disc at case number five, which indicates that a mesh independent solution is achieved. However, investigating the contours of velocity magnitude as shown in Figure 4.2, differences in the flow field between case five and six can be located. The arrow in the bottom right corner of the contours indicates the rotation direction of the disc.

The difference can especially be seen at the lower side of the vane where the velocity distribution does not correspond. As seen in Figure 4.2, the computed velocity is higher in case five compared to case six, as well as that the flow out of the disc is not captured properly in case five. The same trend can be observed studying plots of the coefficient of pressure in the same area. Comparing case six to case number seven shows better correlation. Therefore, case number six, which has 1.8 million cells, is chosen as the mesh independent solution since it provides high accuracy at a reasonable computational cost.

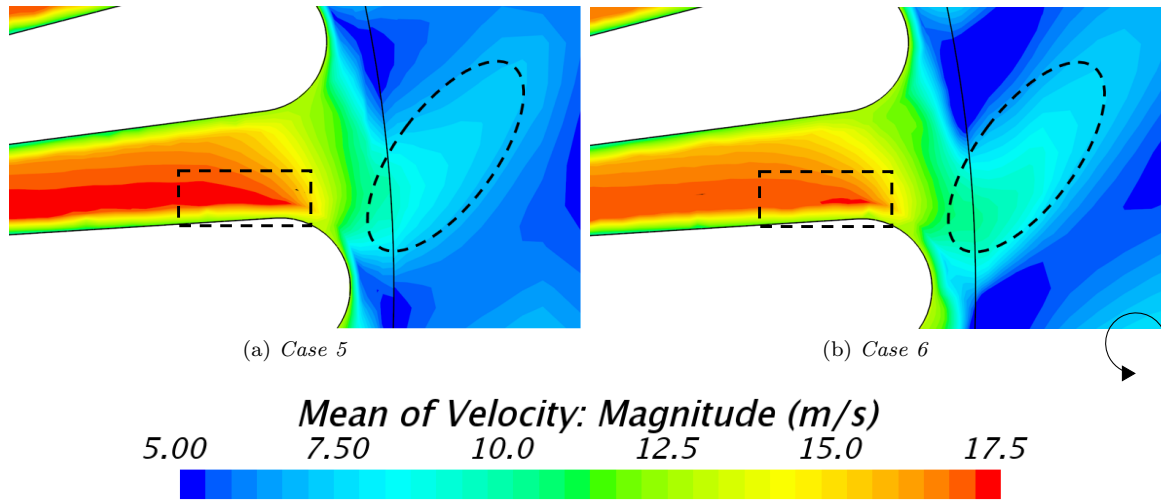


Figure 4.2: Velocity magnitude through a vane in the x - y plane for case five and case six.

4.1.2 Rotation Modeling Techniques

To find a suitable rotation technique, MW, MRF and RBM were run according to Section 3.2. The outcome of the simulations is compared using tabulated values and contours as presented in this section. The parameters monitored were mass flow through the vanes and the computational run-time required to achieve convergence. Table 4.1 illustrates the parameters for simulations with and without solving the energy equation, i.e. with heat transfer and a disc and domain temperature of 20 °C and without heat transfer.

Table 4.1: Monitored parameters used to compare rotation techniques.

Function	MW	MRF	RBM	MW (with HT)	MRF (with HT)	RBM (with HT)
Mass flow through disc [kg/s]	$6.00 \cdot 10^{-3}$	$4.51 \cdot 10^{-2}$	$4.71 \cdot 10^{-2}$	$6.34 \cdot 10^{-3}$	$4.56 \cdot 10^{-2}$	$4.75 \cdot 10^{-2}$
Core hours [h]	7	9	102	9	10	115

From the table, it can be noted that the mass flow through the disc between MRF and RBM are comparable. However, the mass flow predicted using moving wall method results in a much lower prediction of the mass flow compared to the prediction using MRF and RBM. To investigate the HTC prediction for the different rotation techniques, they were compared with a disc temperature simulating a hot brake disc of 520 °C. Table 4.2 presents the Specified Temperature HTC from the brake disc for the three different rotation techniques for both the vanes and friction surfaces.

Table 4.2: Monitored heat transfer parameters used to compare rotation techniques.

Function	MW (with HT)	MRF (with HT)	RBM (with HT)
Specified Temperature HTC Friction Surfaces [$W/m^2 K$]	21.3	29.6	29.0
Specified Temperature HTC Vanes [$W/m^2 K$]	14.9	49.4	55.7

Once again, the predictions for MRF and RBM correlate fairly well, whilst MW is far from these values. The reason for the Specified Temperature HTC to be slightly higher on the vanes for RBM can likely be due to that the computed mass flow through the disc for RBM is slightly higher as seen in Table 4.1. Comparing the achieved values for MRF and RBM to previous findings shows that they are in good correlation, indicating that these rotation methods are applicable, as confirmed by other investigations [1]. To further investigate why

MW predicts the heat transfer to be lower and the mass flow to be almost ten times lower compared to MRF and RBM, the velocity vectors in the vanes for moving wall and MRF are compared as shown in Figure 4.3.

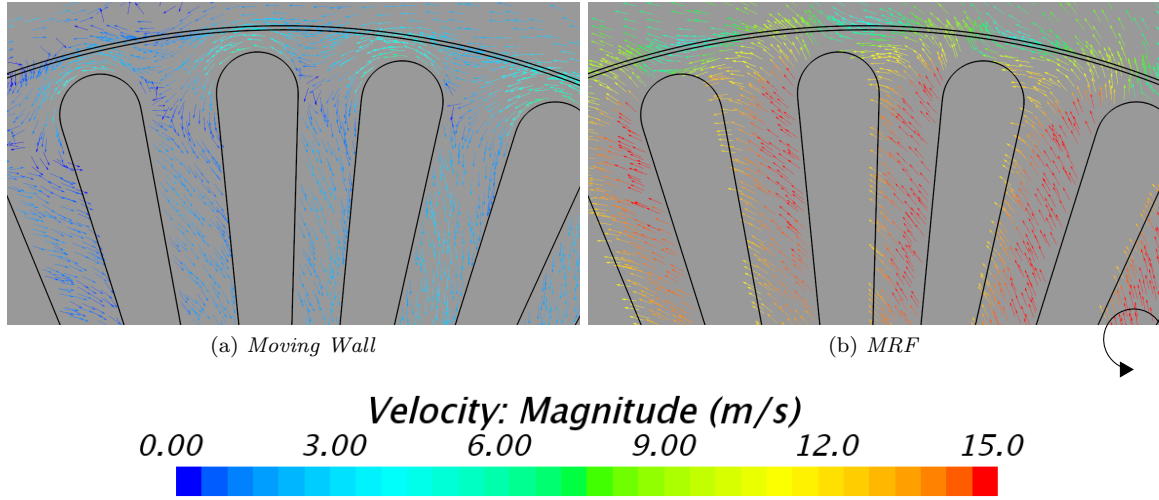


Figure 4.3: Velocity vectors colored with velocity magnitude through a vane in the x - y plane for moving wall and MRF.

As can be observed by the direction of the vectors, MRF captures the expected pumping effect, whilst moving wall does not capture it. The reason for moving wall to predict an almost 10 time lower mass flow can be that in moving wall, only a tangential velocity is applied to the rotating surface, whilst the fluid region in the vanes is not given a velocity, as is the case for MRF and RBM. Not only does this result in a low velocity in the vanes but also in flow that streams in the opposite direction to what is expected, thereby being unsuitable for this study.

To investigate if there are differences in the flow field between MRF and RBM, contours of pressure coefficient, vectors coloured with the velocity magnitude and the specified temperature HTC are investigated. In Figure 4.4, the velocity magnitude is displayed. For MRF, the solution is averaged over 1000 iterations whilst for RBM the solution is averaged for one full rotation after having achieved a converged solution.

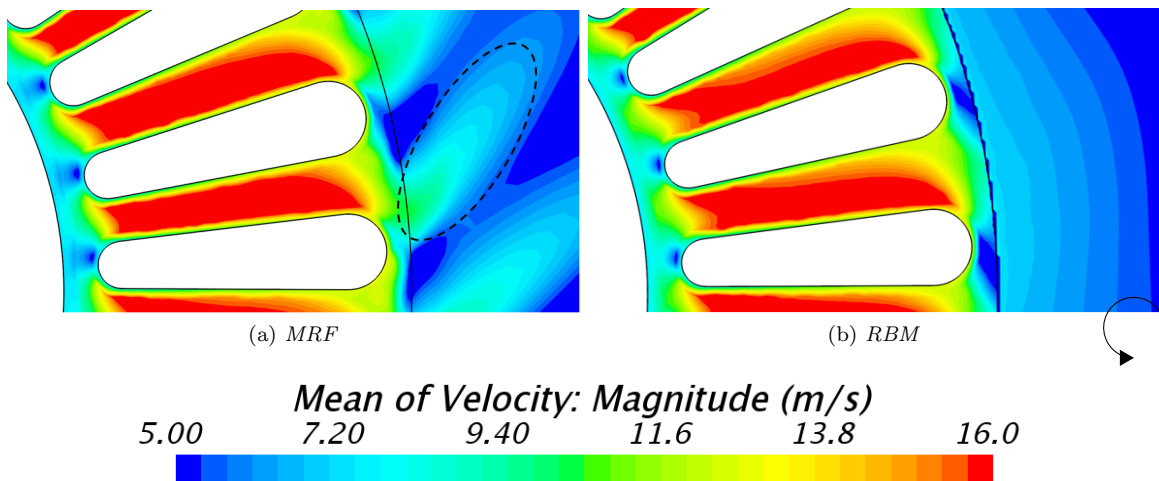


Figure 4.4: Velocity magnitude through a vane in the x - y plane for MRF and RBM.

As can be seen from the flow exiting the vane, unrealistic exit jets are formed in the case simulated with MRF. For the case simulated with RBM these distinct exit jets are not formed, instead a more realistic smooth exit

flow is formed. The reason for this is that in RBM, the location of the vanes are moved, meaning that as they rotate, the exiting flow will also rotate and move through the domain. For MRF, the rotation is only simulated by adding a grid flux of volume flow and body forces, meaning that the vanes, nor the exit jets are moved, whereby the smooth flow cannot be achieved. It is also important to note that the dark blue coloured step pattern for RBM, indicating low velocity, located at the outlet is due to the averaging within the first cell, as a consequence of the type of representation applied.

Investigating the computational time, Table 4.1, showed that the time required to achieve convergence using RBM is about 10 times as high compared to MW and MRF. This is unsurprising since, RBM cannot be applied in steady-state and instead requires a unsteady set-up. It can also be noted that a simulation running with MRF, including solving of heat transfer, i.e. with energy equation turned on, required only 10 % more run-time. However, this time-reduction may not be a general observation for solving without the energy equation, since the temperature in this case was the same as the ambient temperature, whereby no heat transfer will occur between the disc and domain.

4.1.3 Turbulence Models

As presented in the theory section about turbulence, Section 2.3, the realizable 2-layer $k - \varepsilon$ and SST $k - \omega$ turbulence model have different advantages and disadvantages. Adding that both have been applied to similar studies [3, 10, 27], it is hard to say which will perform better for this study.

The turbulence model comparison was run according to a similar methodology to the rotation modeling techniques, but at a temperature of 520 °C. Once again, contours were looked into and the parameters of interest studied, are as shown in Table 4.3.

Table 4.3: Monitored parameters used to compare turbulence models.

Parameter	$k - \varepsilon$	$k - \omega$
Mass flow through disc [kg/s]	$2.98 \cdot 10^{-2}$	$2.16 \cdot 10^{-2}$
Surface Averaged Specified Temperature HTC [W/m^2K]	36.3	32.1
Surface Averaged Virtual Local HTC [W/m^2K]	62.1	681.6
Core hours [h]	13	17

As can be seen, the mass flow differs with almost 30 % between the two turbulence models. To understand why, contours of the pressure coefficient in the vanes were investigated, displayed in Figure 4.5

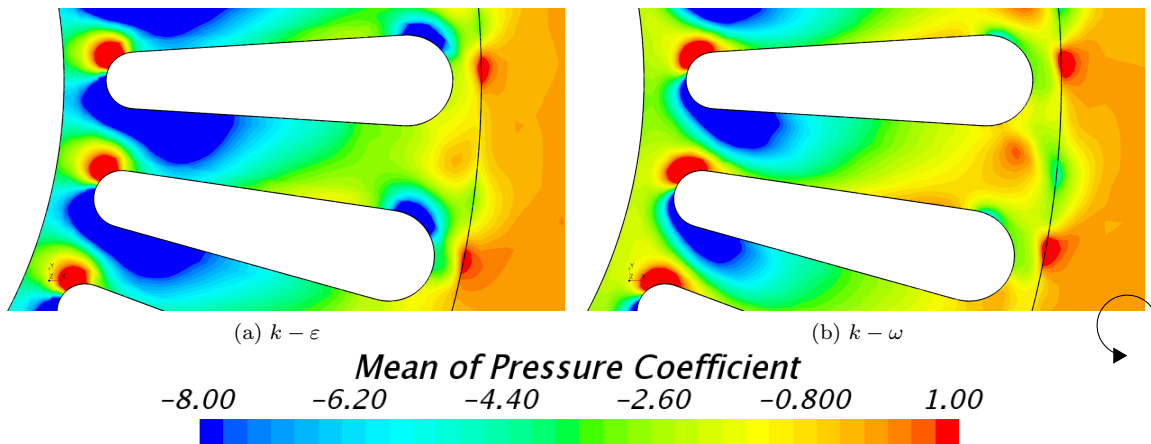


Figure 4.5: Distribution of the pressure coefficient through the vanes in the x - y plane for the turbulence model comparison.

From Figure 4.5, it can be noted that the pressure is computed to be higher, both at the inlet and inside of the vane for the $k - \omega$ turbulence model. This matched with the observations of a lower velocity distribution through the vanes and is likely the explanation of why the mass flow is lower for the $k - \omega$ turbulence model. Even though the effect on mass flow is a decrease of almost 30 %, the heat flux prediction is only decreased around 10 %.

One downside worth mentioning for the $k - \omega$ model is that it cannot accurately estimate the Virtual Local HTC due to the fact that the model requires a low $y+$ value. A low $y+$ value will produce a high reference temperature, since the reference temperature will be chosen as a temperature close to the wall, as well as that the wall function might have problems accurately estimating the reference temperature. Due to these reasons, the Virtual Local HTC will predict high and possibly even erroneous values not accurately resembling the actual heat transfer, as explained in Section 2.5.2 and confirmed by the values in Table 4.3. This suggests that better predictions of Virtual Local HTC can be achieved using $k - \varepsilon$ turbulence model when compared to SST Menter $k - \omega$ turbulence model.

For unsteady simulations, two other turbulence models i.e., IDDES and URANS were compared. For the IDDES, EB $k - \varepsilon$ Detached Eddy Simulation was used and for the URANS, Realizable two layer $k - \varepsilon$. It was found that the average estimation for velocity and mass flow in the case of IDDES was 8% and 14.5% higher than that of URANS. It can also be noted that IDDES takes 8-10% higher computational time compared to URANS and that the IDDES captures the gradients and the eddies in a better resolution as compared to URANS, as can be seen from Figure 4.6, where the light blue area around the disc is more widespread.

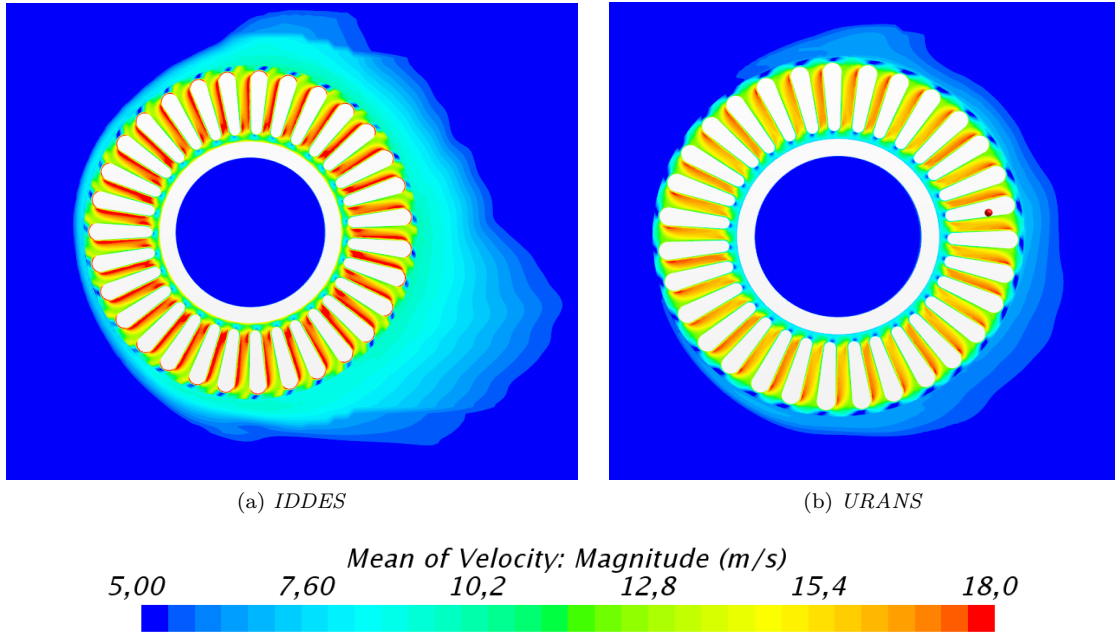


Figure 4.6: *Averaged Velocity Magnitude contour through the vanes in the x-y plane for the unsteady turbulence model comparison, rotation in anticlockwise direction.*

The use of Detached Eddy Simulation resulted in better resolution of the results at a higher computational cost. The major portion of this work uses a steady state approach, hence, considering the computational time of the simulations, the $k - \varepsilon$ turbulence model proved to be more practically advantageous for steady-state simulations and URANS seemed more advantageous in terms of computational time for unsteady simulations.

Having the advantages of lower computational time and easier convergence compared to the $k - \omega$ turbulence model, led to the decision of using the Realizable Two-Layer $k - \varepsilon$ for further studies in this work.

4.2 Heat Transfer Coefficients

To understand how the HTC's should be defined, all four definitions presented in Section 2.5 were compared. Firstly, the results of the first cell height dependent HTC's will be presented followed by the first cell height independent HTC's. The section will end with presenting the results that could result in less time-consuming estimations, i.e. if the Virtual Local HTC can be used for HTC predictions without solving the energy equation and if the HTC is linear as a function of rotational velocity.

4.2.1 First Cell Height Dependent HTC

The averaged HTC values for the friction surfaces of the baseline configuration brake disc are displayed in Table 4.4 based on a varying first cell height. Note that these HTC values are for the friction surfaces only. The reason being that generating such high $y+$ values as in Case 3, decreased the accuracy in the vanes because of too few cells across the boundaries.

Table 4.4: First cell height dependent HTC comparison.

Case No.	First Cell Height [mm]	$y+$ [-]	Virtual Local HTC [W/m^2K]	Local HTC [W/m^2K]	Percentage Difference [%]
Case 1	1.9	15.0	66.9	66.1	1.2
Case 2	2.7	22.3	57.5	56.9	1.1
Case 3	4.2	31.8	51.9	50.8	2.1

As can be seen, the Virtual Local HTC correlates well with the Local HTC and the deviation between the two is approximately two percentage. Another observation is that both field functions show a first cell height dependency. This correlates with literature findings and as explained in Section 2.5, the reason for this is that these functions use the temperature in the cell closest to the wall. At a low first cell height, the reference temperature will be close to the wall temperature, resulting in a low delta temperature, thereby predicting high HTC values, according to equation 2.13 and equation 2.15. Comparing the HTC values to studies with similar objectives show that the obtained HTC values are high compared to commonly estimated HTC values, indicating that these HTC measures are not commonly applied [10, 28, 29].

A way to predict text-book HTC values with these functions would be to increase the $y+$ value, whereby the reference temperature and HTC prediction is decreased. On the exterior surfaces of the brake disc, this can easily be achieved. However, in the vanes this is not possible since the distance between the vane-walls is limited, resulting in that the prism layers would need to be high and the resolution of the mesh would become too low to capture the flow properties properly. Due to the fact that the $y+$ values will be different on the vanes and exterior surfaces, the reference temperature will also vary for different regions.

Another consideration is that the Local and Virtual Local HTC apply wall functions to estimate the reference temperature. Guidelines state that when strong body forces are presented, e.g. in the vanes of a rotating disc, the wall function may reach their limit of predicting accurate solutions [24]. This raises the question of how good the predictions of the Local HTC and Virtual Local HTC are, as will be discussed more in Section 4.2.3.

4.2.2 Non-First Cell Height Dependent HTC

As can be seen in Table 4.5, comparing the Specified Temperature HTC with the Specified $y+$ HTC yielded results that the Specified $y+$ HTC predicted slightly higher HTC values compared to the Specified Temperature HTC.

It is important to note that both the HTC's did not show a first cell height dependency. Furthermore, the values estimated are lower compared to the first cell height dependent values, and also compare better with literature findings [10, 28, 29], indicating that similar definitions are applied in similar studies.

The reason for the Specified Temperature and Specified $y+$ HTC to predict lower values compared to the first

Table 4.5: Non-First cell height dependent Heat Transfer coefficient Comparison.

Case No.	First Cell Height [mm]	$y+$ [-]	HTC [W/m^2K]	Specified $y+$ HTC [W/m^2K]
Case 1	1.9	15.0	32.0	38.6
Case 2	2.7	22.3	31.7	38.5
Case 3	4.2	31.8	31.7	37.9

cell height dependent HTC is that the reference temperature will be lower, resulting in a greater temperature gradient and thereby lower HTC values, according to equation 2.13 and equation 2.14.

An important detail worth mentioning is that the Specified $y+$ HTC is $y+$ dependent. Altering the velocity will result in a different wall shear stress, dictated by equation 2.17. As a result of this, the friction velocity used to estimate $y+$, u^* , will also change, given by equation 2.4. According to equation 2.5, this would result in an either increased or decreased $y+$ value. Since the Specified $y+$ HTC takes the reference temperature at a set $y+$ value, the reference temperature will change for different velocities. This introduces an uncertainty comparing cases with different velocities. Due to this reason, and that the Specified Temperature HTC shows good correlation with literature findings, the Specified Temperature HTC is chosen as the main measure to apply in this study.

4.2.3 Virtual Local HTC

If it is possible to apply the Virtual Local HTC to standard aerodynamic simulations in order to compare the performance of different disc designs, it would be highly valuable since this could result in a run-time reduction, due to eliminating the need to solve the energy equation. In Section 4.2.1, it was explained that the Virtual Local HTC is first cell height dependent, and that the results may be questionable due to it utilizing wall functions for estimation of the reference temperature. To find out if it is applicable for early predictions, it was compared to the Specified Temperature HTC of the disc to see if it follows the same trend. If it does, then it can possibly be used to evaluate changes on a percentage level.

As found in the previous section, the magnitude of the Virtual Local HTC is commonly higher than the Specified Temperature HTC. Therefore, in order to compare both of the HTCs, the values were normalized with their respective value at 800 rpm. Figure 4.7 illustrates the percentage increase of Virtual Local HTC for an increased rotational velocity in relation to the Specified Temperature HTC increase of the disc.

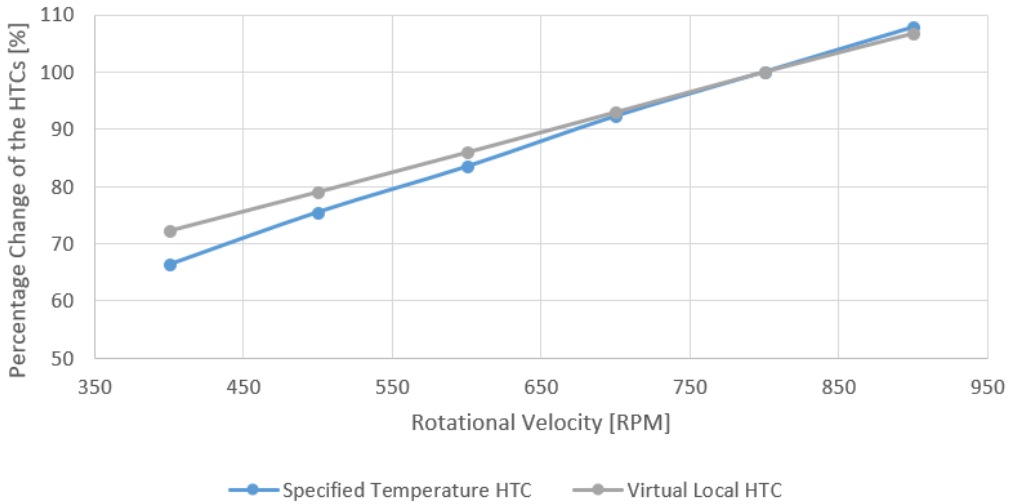


Figure 4.7: Correlation of the normalized Virtual Local HTC and normalized Specified Temperature HTC of the disc as a function of rotational velocity.

It can be seen from Figure 4.7, that the percentage increase of Virtual Local HTC correlates with percentage increase of the Specified Temperature HTC. An important observation to be made, is that the slope of the Virtual Local HTC is less inclined than the slope of the Specified Temperature HTC. In other words, the percentage increase between two rotational velocities is lower for the Virtual Local HTC. E.g. the percentage increase when increasing the rotational velocity from 500 to 600 rpm is 11.0% and 8.7% for the Specified Temperature HTC and Virtual Local HTC respectively. Even though, the increase is not the same, this still indicates that the Virtual Local HTC can be used for early predictions since it estimates a similar trend as the Specified Temperature HTC. Additionally, since the predicted HTC increase is lower, the estimated heat transfer will be lower, thereby over estimating the temperature on the brake disc. In turn, this means that the result can still be used, considering the over estimations as a safety margin in their calculations.

To further investigate the performance of the Virtual Local HTC, three radial brake disc designs with a varying vane width were run with energy on and off, i.e. a standard aerodynamic simulation. Again, the Virtual Local and the Specified Temperature HTC are compared as seen in Figure 4.8.

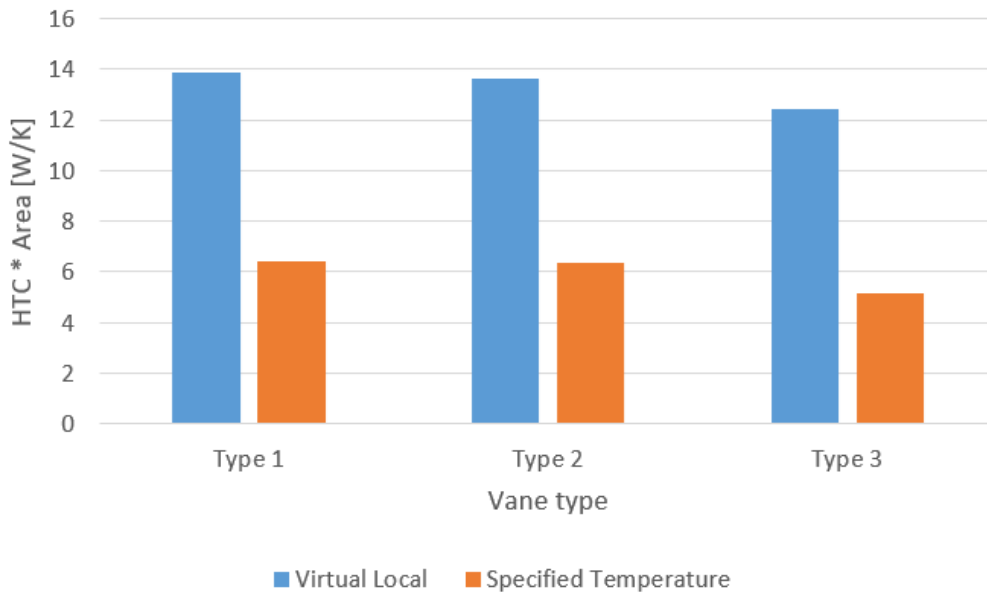


Figure 4.8: Comparison of the Virtual Local and Specified Temperature HTC for three different disc designs

As can be seen from Figure 4.8, the absolute values of the Virtual Local and the Specified Temperature HTC do not correlate well. However, the trend of the Virtual Local HTC follows the same trend as the Specified Temperature HTC, once again indicating that the Virtual Local HTC can be used to estimate what is better or worse, but not how much better or worse.

In addition to this, the computational time was monitored for cases where the Virtual HTC was investigated, i.e. with energy turned off, compared to a case where the Specified Temperature HTC was investigated, i.e. with energy on. For a case with energy off, the simulation was converged in 5 core hours, compared to converging in almost 12 core hours when running with energy on. These time findings, along with the time findings from similar simulations performed with and without solving the energy equation, show that simulations with energy on require an average of 60 % as much run-time for convergence, clearly showing how valuable it can be if the Virtual Local HTC can be used.

Another point to keep in mind is that in simulations that do not solve the energy equation, buoyancy effects cannot be captured. The reason being that there will not be an actual temperature field to affect the buoyancy forces. In Section 4.7, the effect of gravity on natural and forced convection is investigated. It is found that at no rotational velocity, i.e. a natural convection case, the heat transfer requires gravity and a temperature field to be captured, meaning that the Virtual Local HTC cannot resemble the heat transfer for a non-rotating brake disc. However, it is also found that in the range of a rotational velocity of 100-1200 rpm, with the other

inlet parameters kept constant, the effect of gravity is insignificant. Thus, it should not be problematic to use the Virtual Local HTC in this range for percentage performance investigations, even though gravity and the temperature field is not considered.

As shown, the Virtual Local HTC predicts the correct behavior both when the rotational velocity is increased, i.e. when the brake discs mass flow is increased, and when comparing design parameters. This indicates that it can be used as a valid measure to evaluate different brake designs, and also investigate the effect of addition of adjacent parts such as a wheel deflector or a rim, since this in turn will affect the temperature and air flow that the disc is exposed to. Though, it should be kept in mind that applying the Virtual Local HTC comes with some limitations. Firstly, the first cell height, i.e. first prism layer, restricts the use of certain models due to the recommendation of a $y^+ > 30$ to predict a viable reference temperature. Furthermore, the mesh is constrained in such that the first cell height should be kept constant over the object of interest, as well as that it should be the same when comparing different simulations. However, keeping this in mind, the results indicate that the Virtual Local HTC can be used for early predictions in determining the qualitative performance rather than a quantitative performance of a brake disc design.

4.2.4 Linearization and Scalability of HTC

Another observation that can be made from Figure 4.7 is that both the Specified Temperature and Virtual Local HTC dependency on rotational velocity follows a linear trend. Commonly, the HTC is assumed to be linearly dependent on the rotational velocity and temperature for forced convection cases in order to reduce the number of simulation to run. Hence, regions of the disc were studied for a linearly increasing HTC.

Figure 4.9 shows the Specified Temperature HTC as a function of rotational velocity at a constant temperature of $520\text{ }^{\circ}\text{C}$. Here the gradient between 400 and 900 rpm was calculated, and then used to extrapolate values of the HTC at 200 and 1600 rpm for both the vanes and friction surfaces. The lines with circles show extrapolated HTC values whilst the circular points indicate the values obtained from simulations. To verify the extrapolations at 200 and 1600 rpm, simulations were also run at these velocities and from which the HTC based on the simulation was added to the graph.

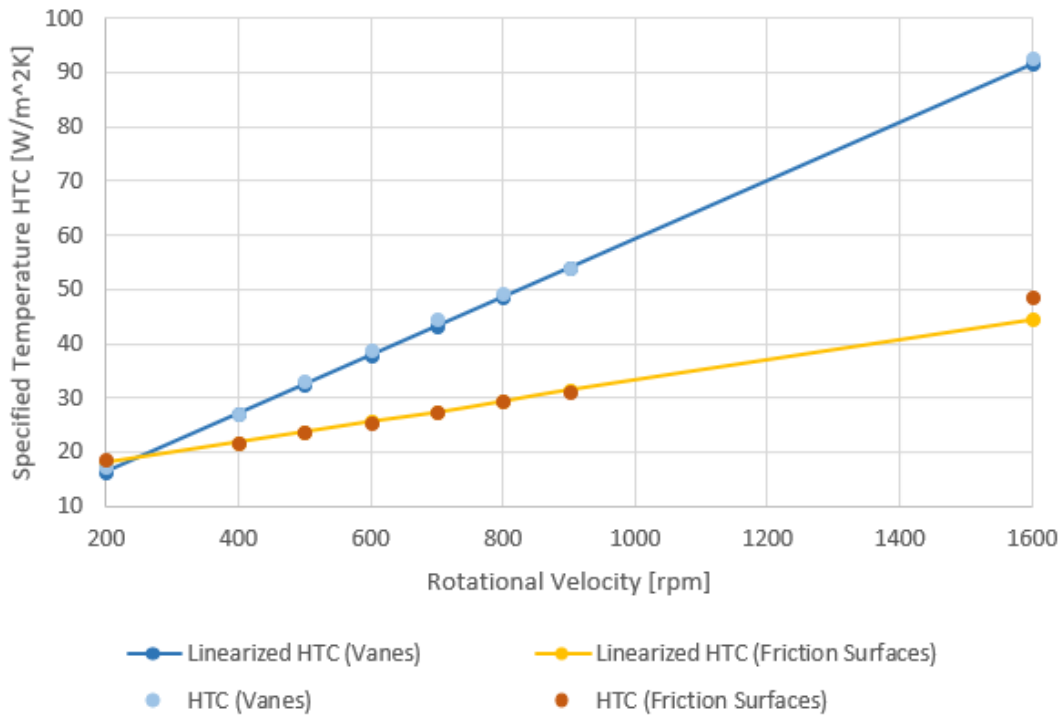


Figure 4.9: Results from investigation of linear behavior of Specified Temperature HTC. Lines extrapolated based on HTC values between 400-900 rpm.

The extrapolated results are in good correlation with the actual simulation results, especially at the lower rotational velocities, where the correlation is 4 %. At higher rotational velocities a deviation of up to 9 % can be found.

Similarly, the linearity of the Specified Temperature HTC on temperature was investigated, this time at a constant rotational velocity of 800 rpm. To linearize the HTC, the gradient was calculated for the slope between 420-520 °C. In Figure 4.10, The line with circular markers indicate the linearized Specified Temperature, whilst the circular markers indicate the Specified Temperature HTC values obtained from simulations.

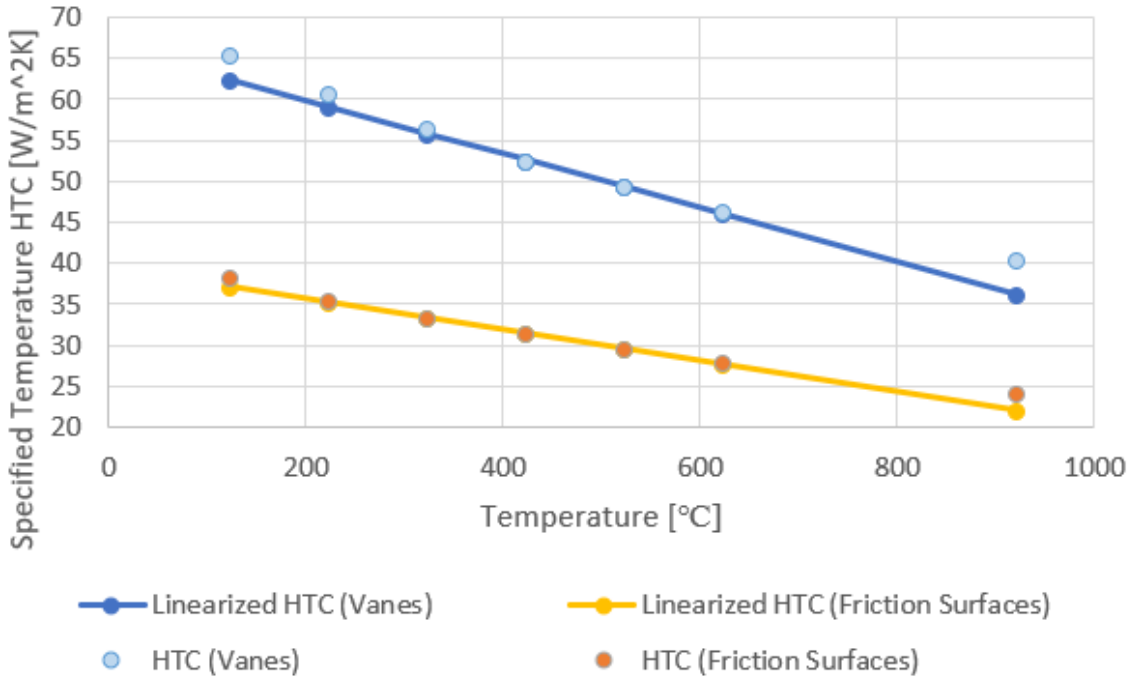


Figure 4.10: Results from investigation of linear behavior of Specified Temperature HTC. Lines extrapolated based on HTC values between 420-520 °C.

As can be seen in Figure 4.10, the linearized prediction correlates well close to the range 420-520 °C. However, deviating further from that temperature range, i.e. to 120 or 920 °C results in a deviation of up to 10%. Nonetheless, note that the predicted linearized HTC in both Figure 4.9 and Figure 4.10 is lower than the Specified Temperature HTC, meaning a higher temperature prediction of the disc. This can be beneficial as a higher prediction can account for a safety margin. However, the assumption of an entirely linear HTC is clearly, not entirely correct. A better approximation would be to run at least three points and calculate a temperature dependent HTC function based on this. To investigate if a temperature dependent HTC curve for the whole disc is scalable based on the rotational velocity, a function for the HTC of the disc was calculated based on the same temperatures as shown in Figure 4.10, at a rotational velocity of 800 rpm. This resulted in a curve described by equation 4.1.

$$HTC_{800rpm}(T) = -1657 \cdot T^{0.008757} + 1793 \quad (4.1)$$

To investigate if this curve is scalable based on rotational velocity, three additional simulations at 120, 520 and 920 °C were run for both 400 and 1200 rpm. The respective HTC values were then divided by the values at 800 rpm which resulted in an average scaling factor of 0.646 and 1.335 for 400 and 1200 rpm respectively. Multiplying this factor with equation 4.1 and plotting the actual simulation values, circular points, to the approximated HTC curves, lines, results in Figure 4.11.

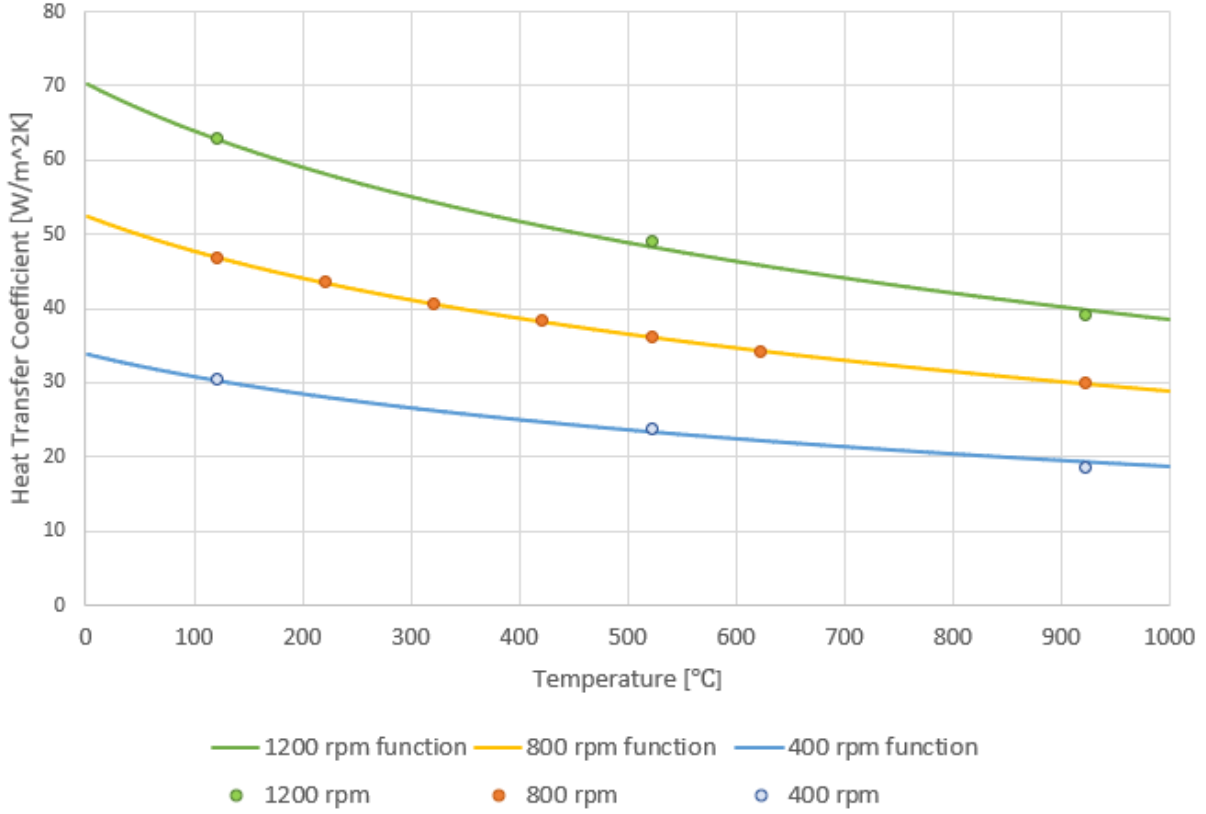


Figure 4.11: *Non-linear functions and values obtained from simulations of the Specified Temperature HTC.*

As can be seen, the curve correlates well with simulation values. This indicates that a good trade-off between accuracy and simulation time is to generate a HTC function dependent on temperature, and then scale it for different rotational velocities. Instead of having to run 5-7 simulations steady-state simulations for each rotational velocity, it is enough to run 5-7 simulations for one rotational velocity, and then an additional 2-3 simulations for each extra rotational velocity to be studied. Another possibility would be to run a unsteady simulation since the disc will be cooled, whereby the HTC will be solved as a function of temperature. However, as explained in the Rotation Technique section, section 4.1.2, the computational time required for a unsteady simulation was shown to 10 times as high, thus much more computationally expensive.

Investigating Figure 4.9 and Figure 4.10 may raise the question of why the slope is steeper for the HTC of the vanes compared to the slope of the HTC of the friction surfaces. The reason for this is that the vanes are more affected by the increased rotational velocity compared to the friction surfaces, which will be discussed more in Section 4.3.2.

4.3 Effect of Parameter Variations on Heat Transfer

To further understand heat transfer and the flow field from and around brake discs, inlet parameters were altered. Namely, the heat transfer development of the friction surfaces and vanes was studied as the domain inlet velocity, disc rotational velocity, ambient temperature and disc temperature were varied.

4.3.1 Ambient Velocity

In Figure 4.12, the percentage change for the heat transfer is presented for the vanes and friction surfaces as well as the percentage change in mass flow as a function of the free stream velocity. The disc rotation velocity and temperature were kept constant at 800 rpm and 520 °C respectively.

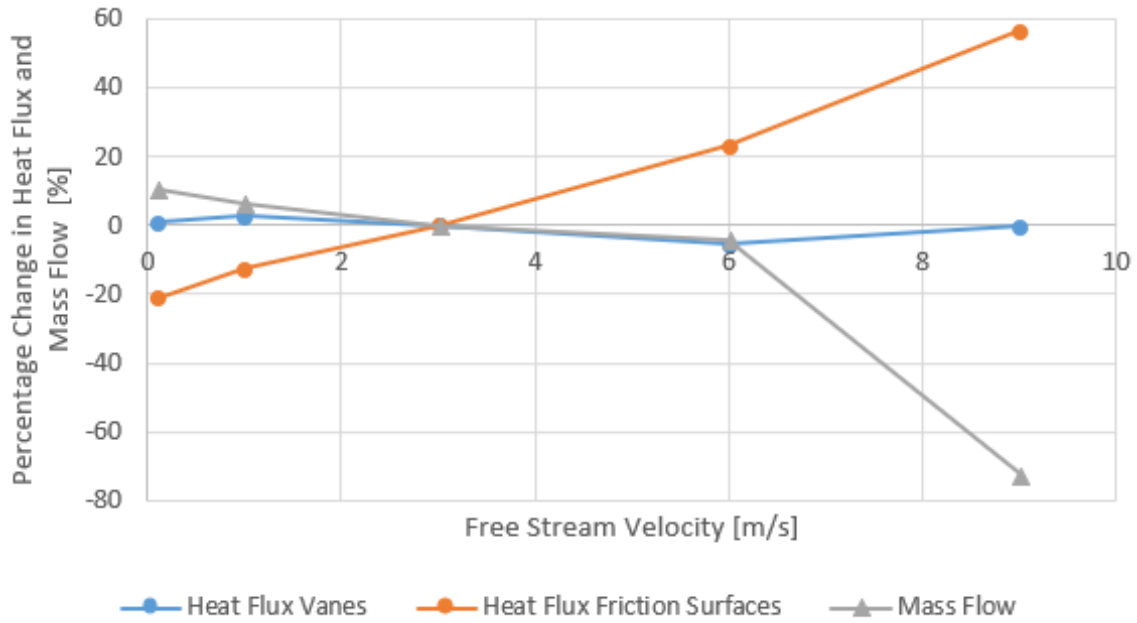


Figure 4.12: *Effect of an increase in free stream velocity on heat flux and mass flow through the vanes.*

As can be observed, the change in free stream velocity has a low impact on the heat transfer of the vanes, which fluctuates in the range of 2700-2900 Watts. On the contrary, the friction surfaces show a significant increase from 1200 to 2300 Watts, for an increase in free stream velocity, correlating well with literature findings [21]. The main contributor to this increase is that the friction surfaces will be exposed to an increased mass flow of air, whereby they can dissipate more heat. The mass flow shows a continuous drop for an increased free stream velocity. To investigate why mass flow through the disc is decreased, velocity vectors on the outlet of the disc facing the oncoming flow are studied, Figure 4.13.

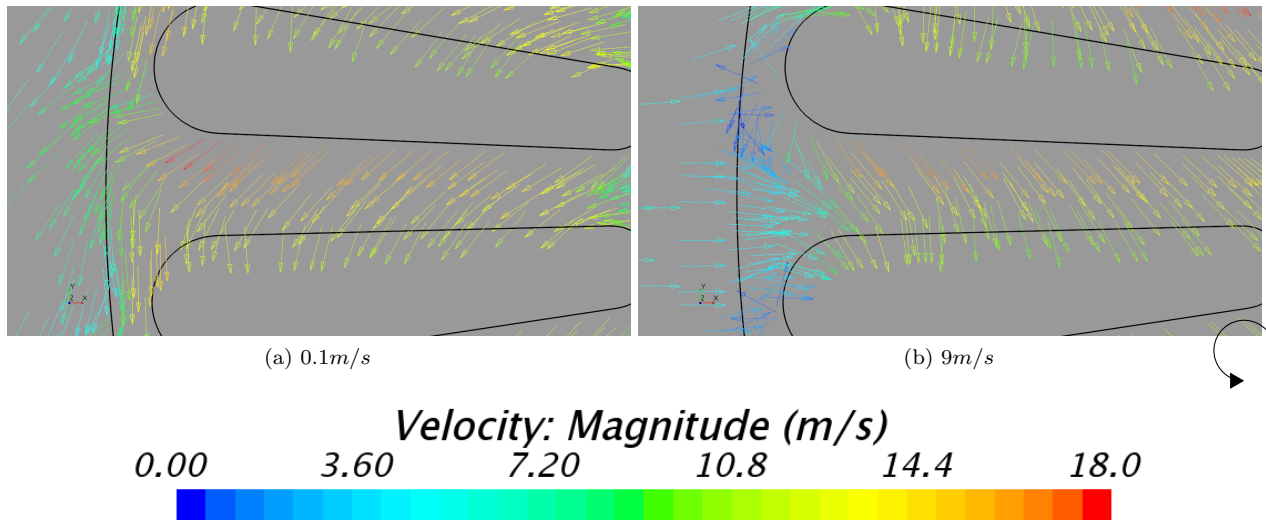


Figure 4.13: *Velocity vectors in the x-y plane on the outlet of the vanes facing the oncoming air.*

As can be seen, at a high oncoming velocity, the flow in the stagnation area cannot exit the disc at the outlet. This is because the oncoming flow reaches the vane outlet at a higher velocity compared to the pumped flow. Already for an oncoming velocity of 9 m/s, it can be shown that the flow forces its way through the outlet of the brake disc, Figure 4.13. Similar studies have also shown this effect which is due to that the oncoming flow produces a higher force compared to the centrifugal pumping force, in the end resulting in less performance of

the vanes [30]. Although the flow forces its way through the outlet, the performance of the vanes only changes moderately. To understand this behaviour, the velocity vectors are further investigated, as shown in Figure 4.14.

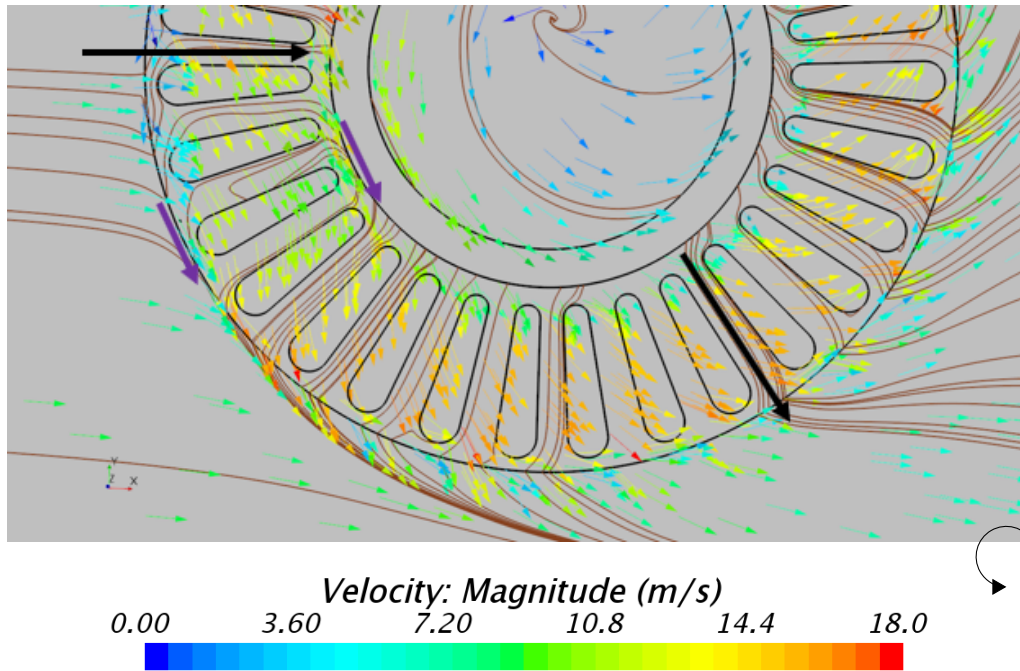


Figure 4.14: *Velocity vectors in the x-y plane coloured with velocity magnitude and streamlines through the vanes.*

From the velocity vectors it can be seen that instead of the flow entering the inlet on the left side of the disc, the oncoming flow, will travel out through the inlet into the hat, i.e. the part of the brake disc attaching to the wheel hub. In the hat, it rotates and then exits the disc on the right side in the correct way, at a higher velocity compared to when the oncoming flow is 0.1 m/s. However, this is not expected to take place if the disc would have been mounted on a car, since the air in the hat is then filled by the wheel hub. Another observation from Figure 4.14, is that in the area indicated by the two purple arrows, the flow travels parallel to the vanes instead of travelling through them. This also happens on the opposite side of the brake disc. To see the effect of this on heat transfer, contours of Specified Temperature HTC are plotted on the interior as can be seen in Figure 4.15.

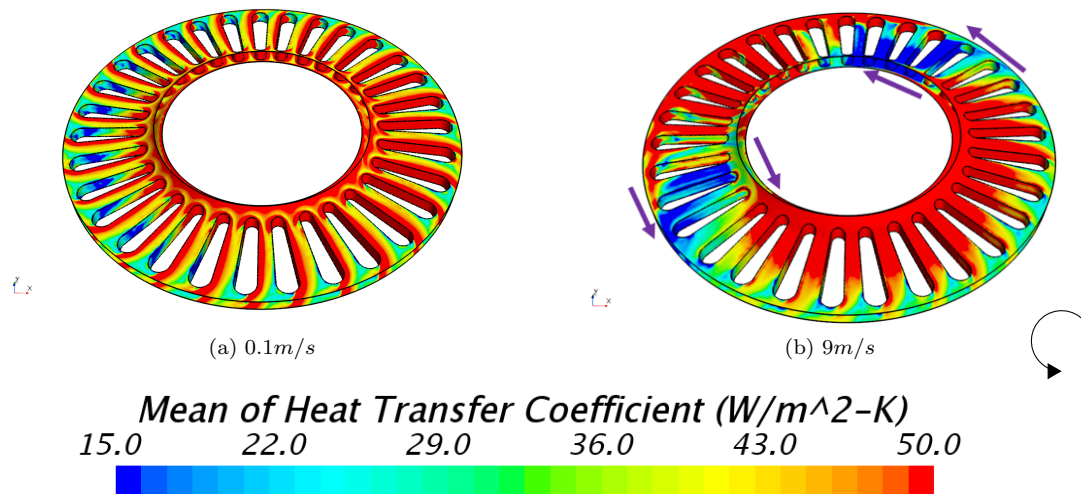


Figure 4.15: *Specified Temperature HTC on the vanes.*

As is expected, the HTC distribution is very even for an oncoming flow of 0.1 m/s. For a high oncoming flow, it is the other way around. In this case, the right area, where the flow is traveling at a higher velocity, has a higher HTC, whilst the areas where the flow travels parallel to the vanes, indicated by the purple arrows, have a lower HTC. This means that at a high oncoming velocity, some areas will transfer more heat and other areas less heat, whilst at a low oncoming velocity, the vanes will transfer heat more uniformly, explaining why the change in oncoming velocity only shows a small effect on the heat transfer performance of the vanes. Nonetheless, this effect is small compared to the heat transfer performance of the friction surfaces, which is why the overall heat transfer performance increases when the oncoming velocity is increased.

However, it is important to remember that in an actual car, the brake disc will be surrounded by adjacent parts such as the calliper, wheel and wheel-house, resulting in that an increase in the vehicle speed will not directly increase the oncoming velocity with the same factor to the disc. Furthermore, if the flow would come at an angle, or a cooling duct would have been present to direct the flow as such in [31], the increase in free stream velocity would most likely result in an increased cooling performance due to the fact that more mass flow would have passed through the vanes.

4.3.2 Disc Rotational Velocity

Altering the rotational velocity of the brake disc showed a different behavior compared to an altered ambient velocity as can be seen Figure 4.16.

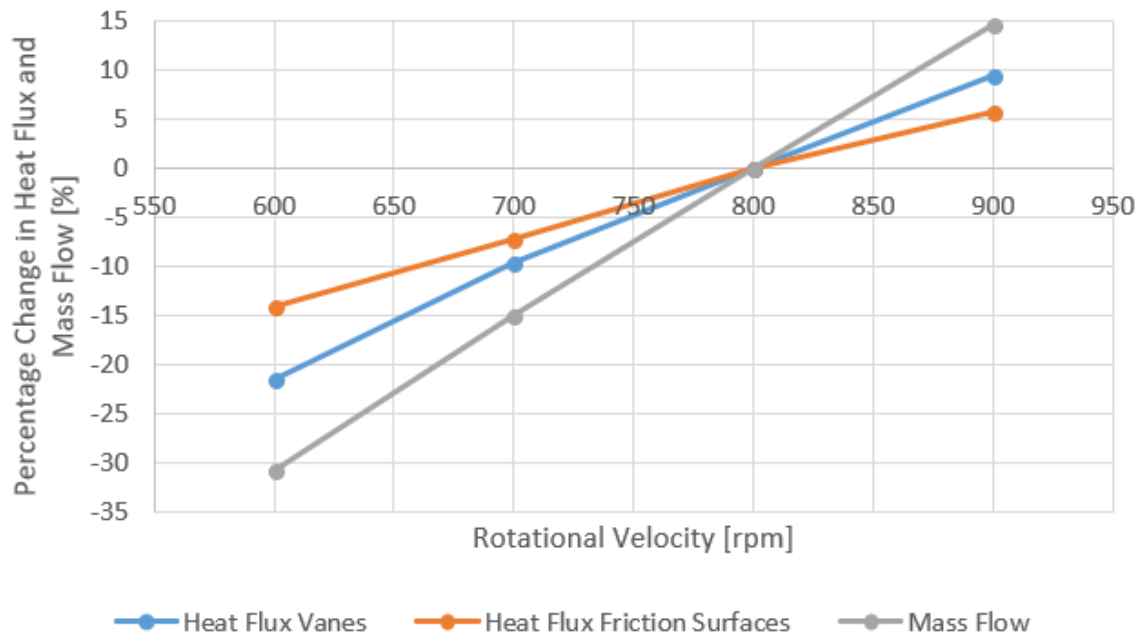


Figure 4.16: *Effect of an increased disc rotational velocity on heat flux and mass flow through the vanes.*

From Figure 4.16 it can be observed that as the rotational velocity is increased, the performance of the vanes is increased more, 2200-3100 Watts, compared to the performance of the friction surfaces, 1300-1600 Watts. Similar trends can also be found in literature [21]. The main reason for this can be explained by investigating the mass flow through the disc as displayed in Figure 4.16. As the rotational velocity is increased, the mass flow of the brake disc is also increased in a linear fashion due to an increased pumping effect of the disc. The increased mass flow means that more heat can be transferred away from the vanes. Similarly, the friction surfaces will have a higher surface velocity as the rotational velocity is increased. This should increase the velocity of the air that has been in contact with the frictions surfaces, and to investigate this the velocity in the domain is investigated 2 mm from the disc as seen in Figure 4.17.

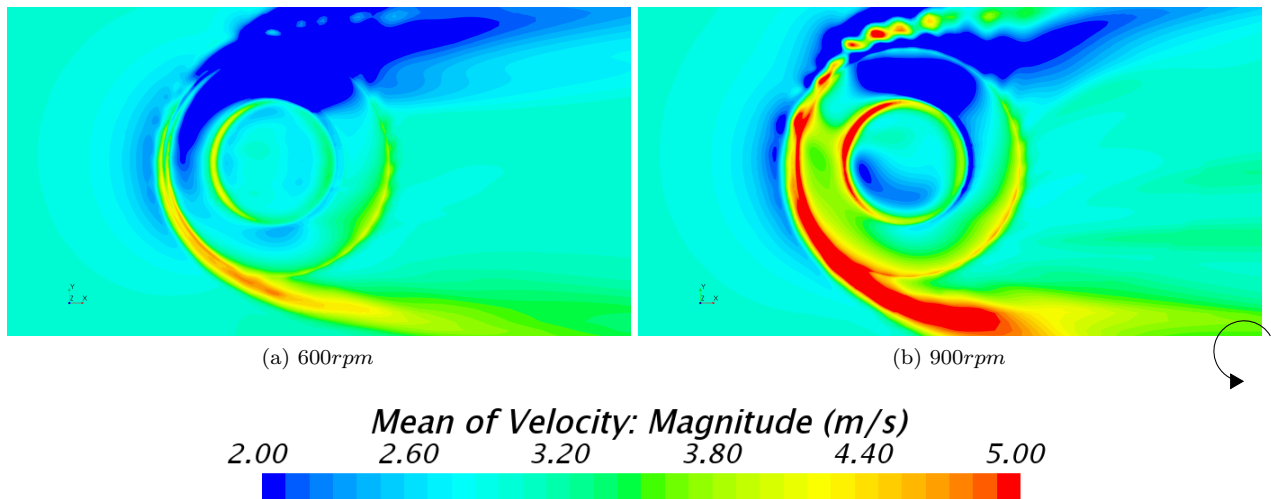


Figure 4.17: Flow velocity on a x - y plane in the domain 2 mm next to the brake disc.

As can be seen, the velocity distribution next to the friction surface of the disc is increased. This will increase the amount of air that interacts with the friction surface whereby more heat can be dissipated from the disc and likely is the explanation to the increased heat transfer from the friction surfaces.

4.3.3 Ambient Temperature

Figure 4.18 presents the percentage change of heat flux and the mass flow as a function of the ambient temperature.

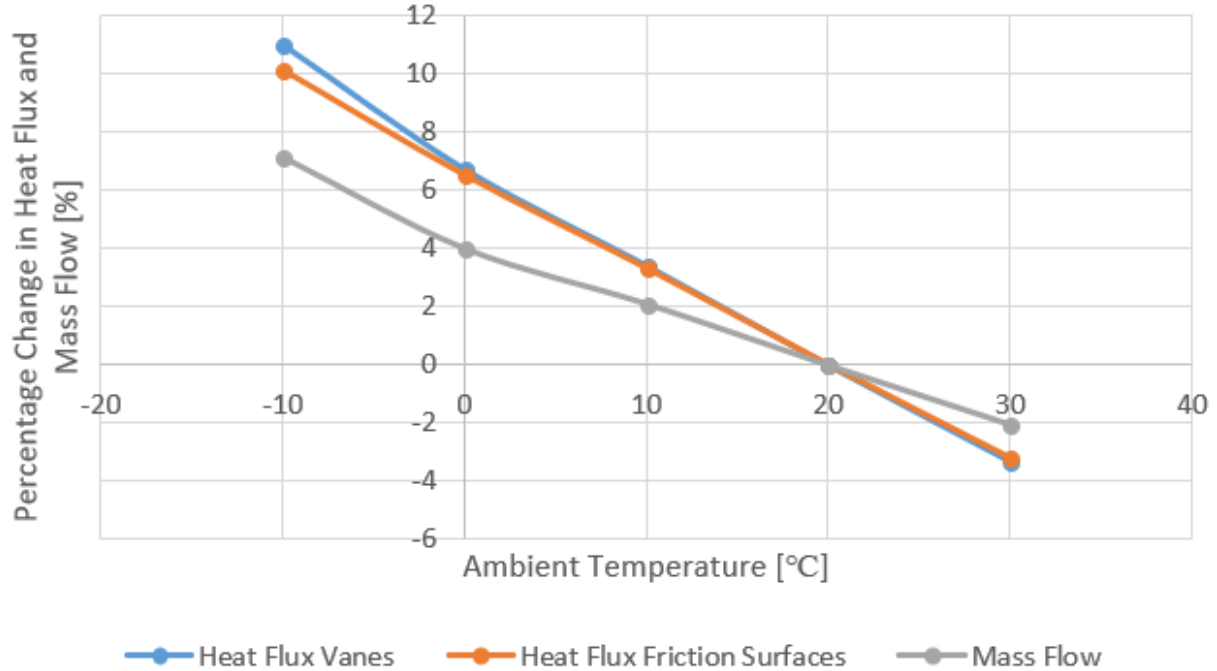


Figure 4.18: Effect of an increased ambient temperature on heat flux and mass flow through the vanes.

As can be observed, a lowered ambient temperature increased the performance of the disc, both for the vanes, 2800-3100 Watts, and for the friction surfaces, 1400-1600 Watts, thereby the overall performance of the disc. This can simply be explained by studying the definition of the convective heat transfer, equation 2.8 in Section 2.4.1, where it states that the heat transfer is dependent on the temperature gradient. When the ambient

temperature is increased and the brake disc temperature is kept constant to 520 °C, the temperature gradient is lowered thereby decreasing the heat transfer from the disc.

Additionally, the mass flow is increased in a linear fashion as the temperature is decreased, which further will increase the heat transfer from the vanes. This increase can be explained by the ideal gas law, according to which, an increased temperature implies a decreased density, equation 4.2.

$$\rho = \frac{p}{RT} \quad (4.2)$$

Since the mass flow is linearly dependent on the density, the increased temperature will thereby also decrease the mass flow, in turn decreasing the heat transfer performance.

4.3.4 Disc Temperature

Similar to the change in ambient temperature, a change in the temperature of the brake disc affected both the performance of the vanes, 2000-3200 Watts, and friction surfaces, 1000-1700 Watts, as can be seen in Figure 4.19.

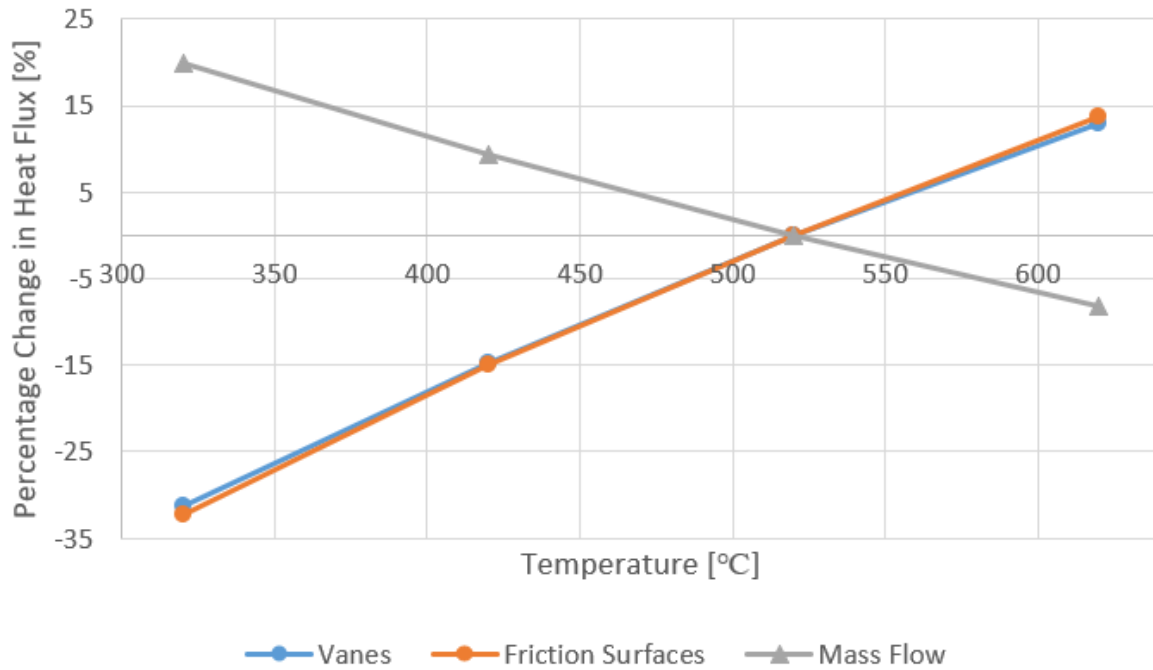


Figure 4.19: *Effect of an increased disc temperature on heat flux and mass flow.*

Once more, the heat transfer behaviour can be explained studying the definition of the convective heat transfer, equation 2.8 in Section 2.4.1. Instead of decreasing the temperature gradient, the temperature gradient is increased by increasing the disc temperature and keeping the ambient temperature constant at 20 °C, whereby also the heat transfer is increased.

Furthermore, a linearly dropping trend for the mass flow can be observed for an increased disc temperature, which once more is dictated by the ideal gas law, equation 4.2, and can be explained by a decreased density. This should result in a decreased heat transfer. However, it is not the case since this effect is lower than heat transfer increase due to the increased temperature gradient.

4.4 Brake Disc Designs

To further understand the flow field and heat transfer of brake discs, typical brake disc geometries were investigated, namely brake discs with radial vanes, tangential vanes and curved vanes. As explained in Section 2.1.2, the vane configuration can have a big impact on the brake disc performance which is why it is important to understand how altering vane parameters can change the cooling performance.

4.4.1 Radial Vane Design

As presented in Section 2.1.2, literature has shown that increasing the vane width can result in improved heat transfer performance [13, 14]. To investigate why, three configurations were investigated with an increasing vane width from a narrow vane to a wide vane, according to Table 3.4, as displayed in Figure 4.21. The percentage change, normalized with the medium sized vanes, on heat transfer, cooling area and mass flow is presented in Figure 4.20.

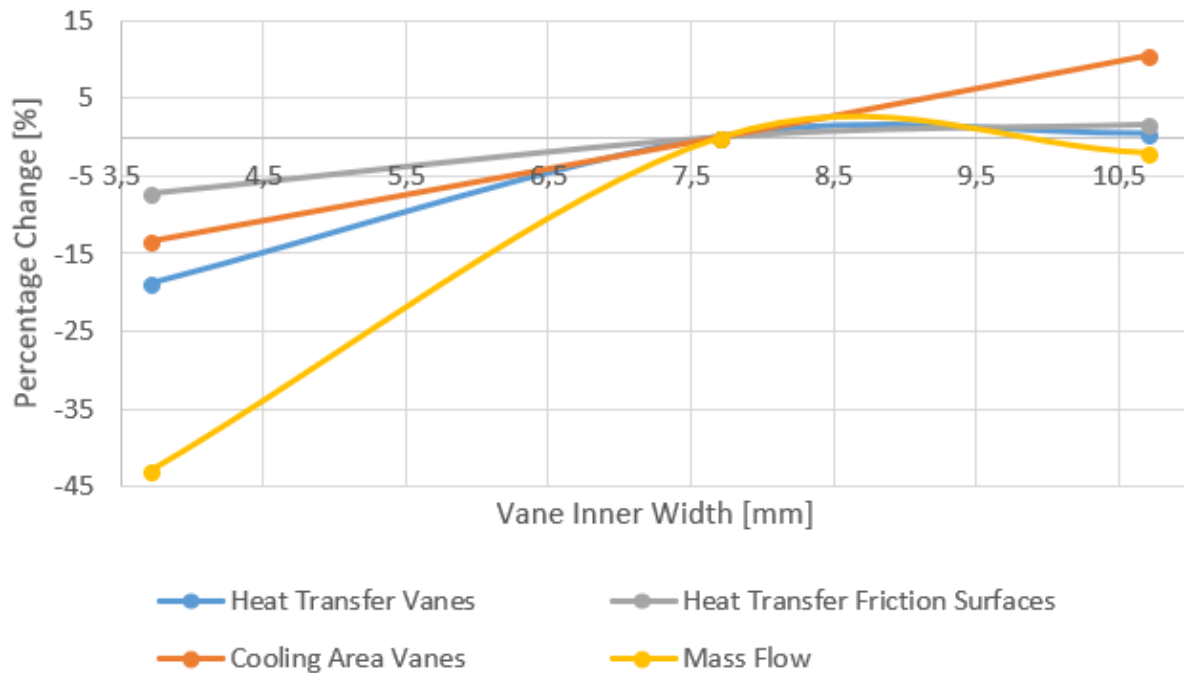


Figure 4.20: *Effect of an decreased vane width.*

As can be seen from Figure 4.20, an increased vane width results in a linearly increasing cooling area, and an increase in heat transfer from both the vanes and friction surfaces. Obviously, the increased vane width will result in wider paths for the air to pass through, resulting in that more air can be pumped through the disc, being one of the reasons for an increased heat transfer performance. Another observation to be made from Figure 4.20 is that the medium and wide vane configurations show quite similar performance, whilst the narrow vane configuration shows a lower performance.

To explain why the performance was similar for the medium and wide sized vane configurations and why the narrow vane configuration showed a decreased performance, the flow field was investigated by studying contours of coefficient of pressure and the velocity distribution as well as the HTC. Figure 4.21 shows the pressure coefficient distribution through the vanes.

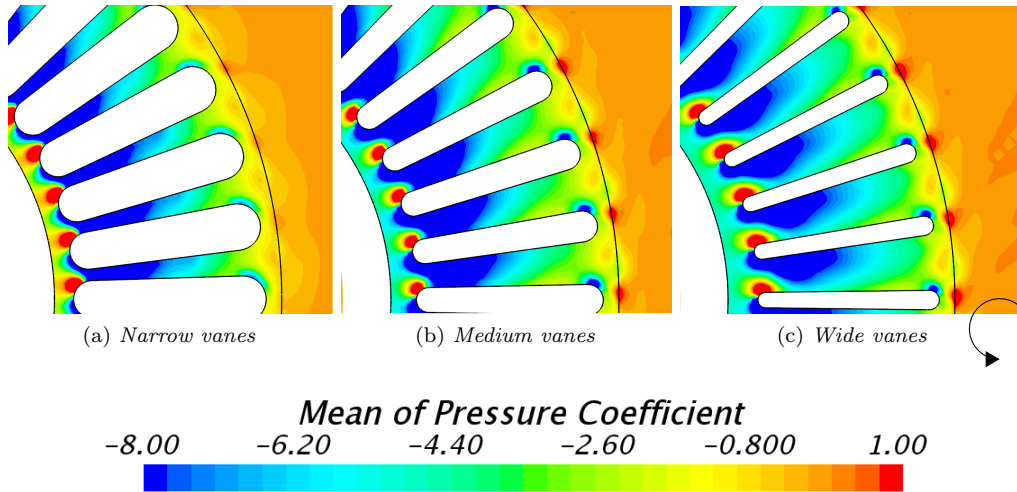


Figure 4.21: *Distribution of the pressure coefficient on a x-y plane through the vanes.*

As can be seen, there is a high pressure area at the leading edge of the vanes. This could indicate a low velocity area and thereby result in a blocking effect of the radial flow. For the wide vane design, the high pressure area does not have a significant effect on the flow, but as the vane width is decreased, the high pressure area clearly starts to interact more with and block the flow. Another observation is that the pressure around the inlet is increased for the narrow vanes compared to the cases for wide vane configuration, which can be due to the restricted flow through the vanes. The high pressure at the inlet will decrease circumferential pressure gradient between the inlet and outlet, thereby being a possible contributing factor to the decreased performance.

Thus, decreasing the vane width from medium-sized to narrow vanes results in a decreased performance, but as can be observed from Figure 4.20, increasing the vane width from medium vanes to wide vanes does not result in a performance increase. To find out why, the velocity magnitude and Specified Temperature HTC are investigated as displayed in Figure 4.22 and Figure 4.23.

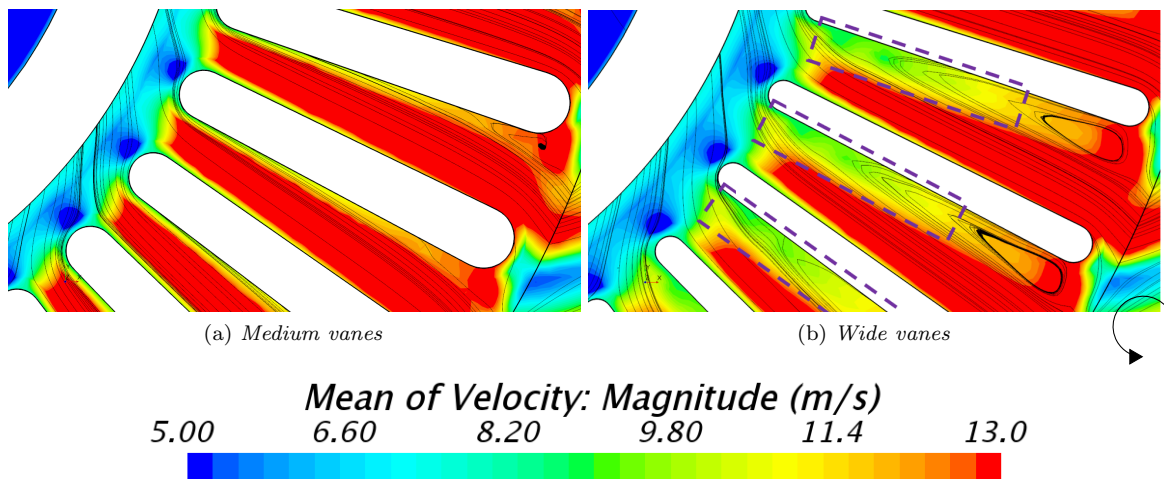


Figure 4.22: *Distribution of the velocity magnitude on a x-y plane through the vanes.*

Investigating the velocity distribution, Figure 4.22, indicating the mass flow, with streamlines shows that there is a more even distribution of mass flow through the medium sized vanes. In case of wide vanes, the streamlines indicate an area of recirculation to the end of the vanes, as well as that the velocity is low on the lower side of the vanes marked by the purple rectangle. Thus, when the width of the vanes is increased, the flow is not forced against the non-attack surface, thereby most likely not interacting with the wall. To find the effect of this on heat transfer, the same areas are displayed with the HTC in Figure 4.21.

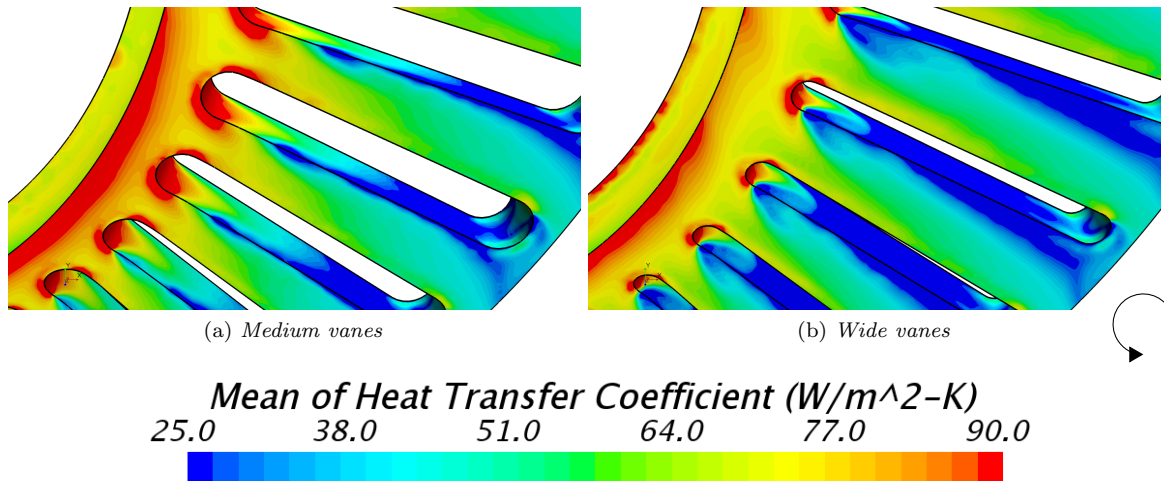


Figure 4.23: Distribution of the Specified Temperature HTC on the vanes in the same area at an angle as Figure 4.22.

Correlating the HTC contour and recirculation areas yields the observations of lower heat transfer in these areas, which likely is the explanation to why the heat transfer is not further increased as the vane width is increased.

4.4.2 Tangential Vane Design

The next design investigated is the tangential brake disc design, which is similar to a radial brake disc design but has the vanes at an angle. In Section 2.1.2, it was explained that changing the vane tangent angle can result in both an improved and decreased heat transfer performance [15, 16]. The vane tangent was varied from 0 degrees, i.e. a radial vane design to 60 degrees with steps of 10 degrees. Once more, the percentage effect of changing the vane tangent, normalized with the best performing tangential set-up, on heat transfer, cooling area and mass flow was studied as can be seen in Figure 4.24.

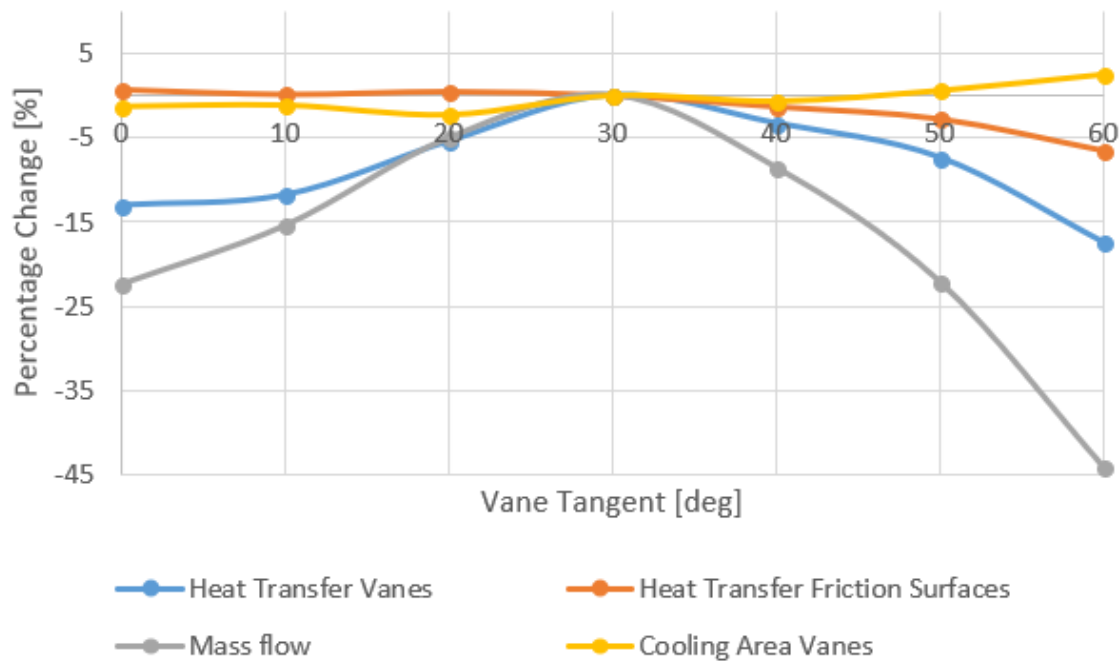


Figure 4.24: Effect of an increase vane tangent angle on heat transfer, mass flow and cooling area.

As expected, the heat transfer and mass flow results for a vane tangent of 0 degrees matched very closely to the results obtained for a radial vane configuration for the same vane thickness. From Figure 4.24 it can be observed that increasing the vane tangent from 0 to 30 degrees yields an improved heat transfer and mass flow performance, whilst decreasing it beyond 30 degrees results in a decreased performance. As for the heat transfer from the friction surfaces and the cooling area, the increased vane tangent does not impact these quantities much. To understand these trends and the flow field, contours of the pressure coefficient distribution, Figure 4.25, and velocity distribution, Figure 4.26 were again investigated. The contours presented below are only for a vane tangent of 30 and 60 degrees. The reason being that the vane tangent of 0 degrees, i.e. a radial vane, showed almost the same results as presented in the previous section for the thin vane configuration, i.e. an uneven velocity distribution with a separation area on the lower side of the vanes.

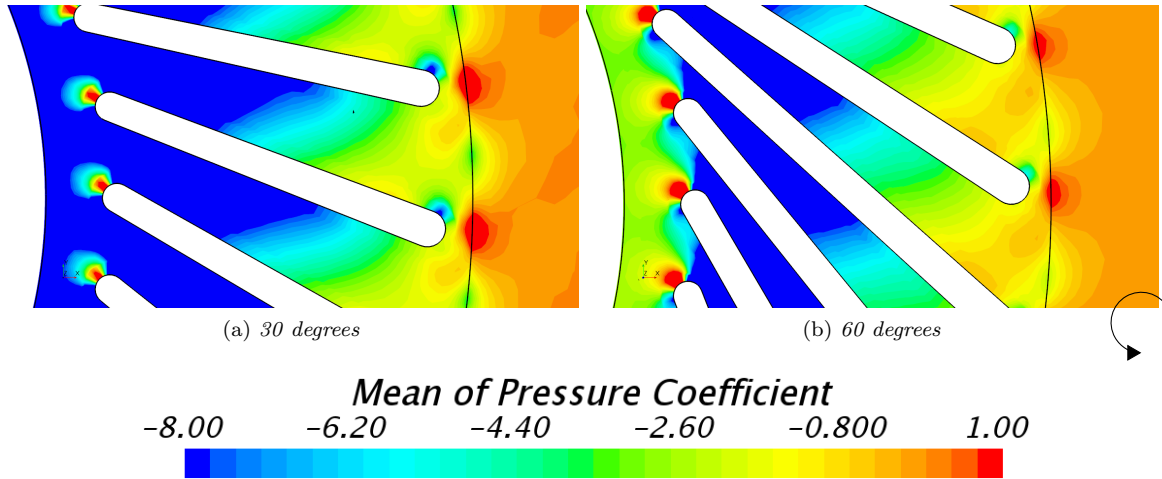


Figure 4.25: *Distribution of the pressure coefficient between the vanes in a x - y plane.*

As can be seen by studying Figure 4.21 and Figure 4.25, as the vane tangent is increased the pressure distribution at the inlet is first decreased as the vane tangent is increased from 0 to 30 degrees, and then again increased as the vane tangent is increased from 30 to 60 degrees. Another observation to be made is that the low pressure region inside of the vanes reaches further to the outlet in the 30 degree case compared to 0 and 60 degrees, as well as that the high pressure area on the vane inlet plays a more significant role in the 0 and 60 degree case. Moving on, the velocity distribution, Figure 4.22 and Figure 4.26, indicates a higher and better distributed mass flow between the vanes for the 30 degree vane tangent case.

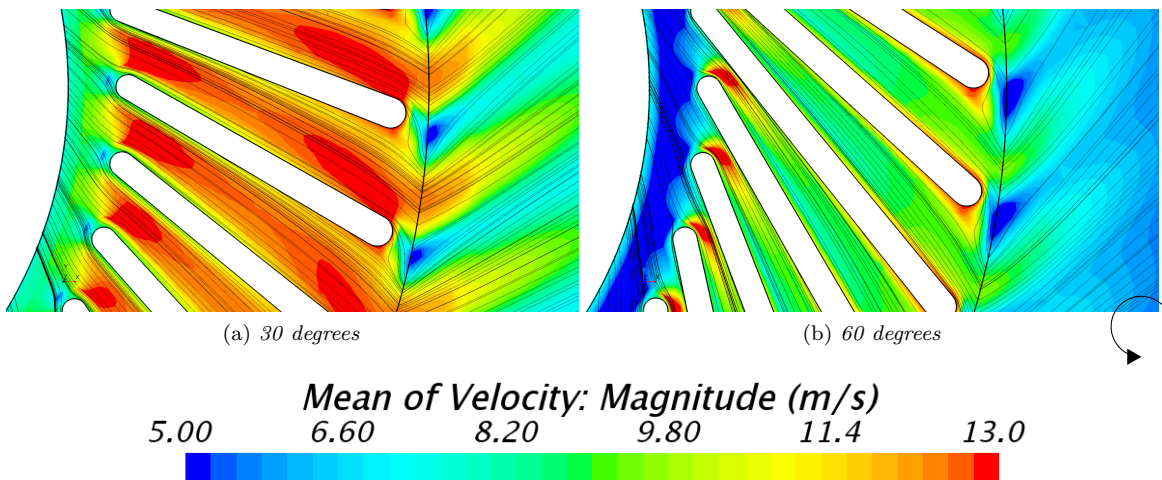


Figure 4.26: *Distribution of the velocity magnitude between the vanes in a x - y plane.*

An explanation for the improved performance could be due to a more favorable inlet angle as the vane tangent was increased to 30 degrees. As can be observed from Figure 4.21 and Figure 4.25, a more favorable inlet angle, i.e. at 30 degrees, results in a high mass flow, indicated by the larger low pressure area at the inlet, thereby improving the pumping effect of the brake disc. As the vane tangent is increased, the gap between the vanes is decreased. The effect of this becomes especially significant increasing the vane tangent beyond 30 degrees. As visualized both by the velocity, Figure 4.26, and the pressure coefficient, Figure 4.25, a high vane tangent results in that less air flow can pass through the disc, resulting in a low velocity region with moderate pressure at the inlet where a low pressure region with high velocity is wanted.

4.4.3 Curved Vane Design

For the curved vane design brake disc, the parameter of interest is the curvature of the vane, which was changed as explained in Section 3.2.4, Table 3.6. Once more, the heat transfer, cooling area and mass flow change, due to altering the vane curvature are studied, as displayed in Figure 4.27. Here the parameter-values are normalized with the maximum vane curvature, i.e. the value at an inner vane point of 30 degrees.

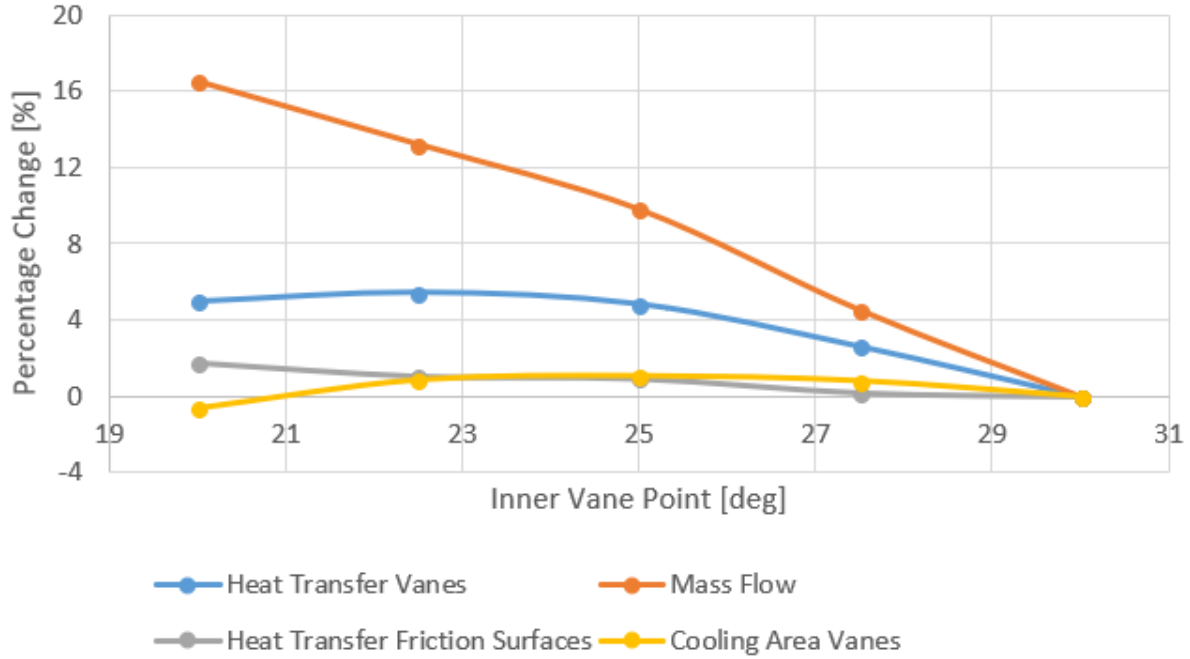


Figure 4.27: *Effect of an increase vane curve.*

From this it is clear that by increasing the vane curvature, the mass flow and heat transfer decrease, whilst the effect on cooling area is found to be insignificant. To explain why this is the case, contours of velocity magnitude, indicating the mass flow, were investigated as presented in Figure 4.28.

As can be seen from the velocity distribution, Figure 4.28, a low curvature generates a suitable inlet angle, prescribing a low pressure area at the inlet and a high mass flow through the vane as indicated by the high and well distributed velocity. When the curvature deviates from this case, similar trends as for the tangential brake disc designs can be observed, i.e. a higher pressure at the inlet and a less uniform mass flow distribution through the vanes.

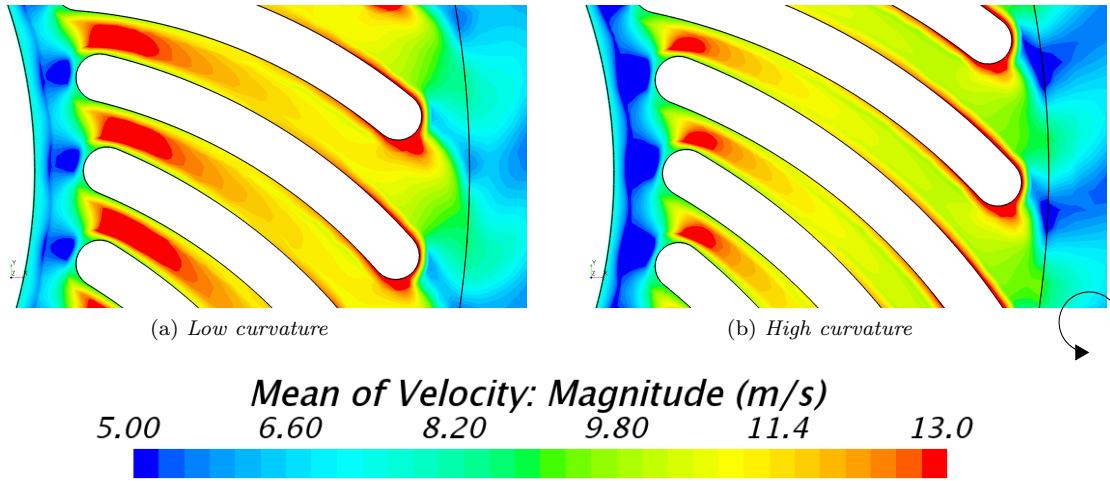


Figure 4.28: Distribution of the velocity magnitude between the vanes.

4.5 Rotation Direction

The cooling performance of different disc types was presented in the above section. In this section, the influence of changing the rotation direction is reported. The radial and tangential vane design were rotated in clockwise direction and anti-clockwise direction based on which the performance was compared. For the investigation, heat transfer and mass flow were monitored as displayed in Figure 4.29. For the radial, as can be expected, changing the rotation direction did not have an influence on the cooling performance and hence have not been reported in this section. In the case of tangential, the performance decrease due to changing the rotation direction from anti-clockwise to clockwise direction is displayed in Figure 4.29.

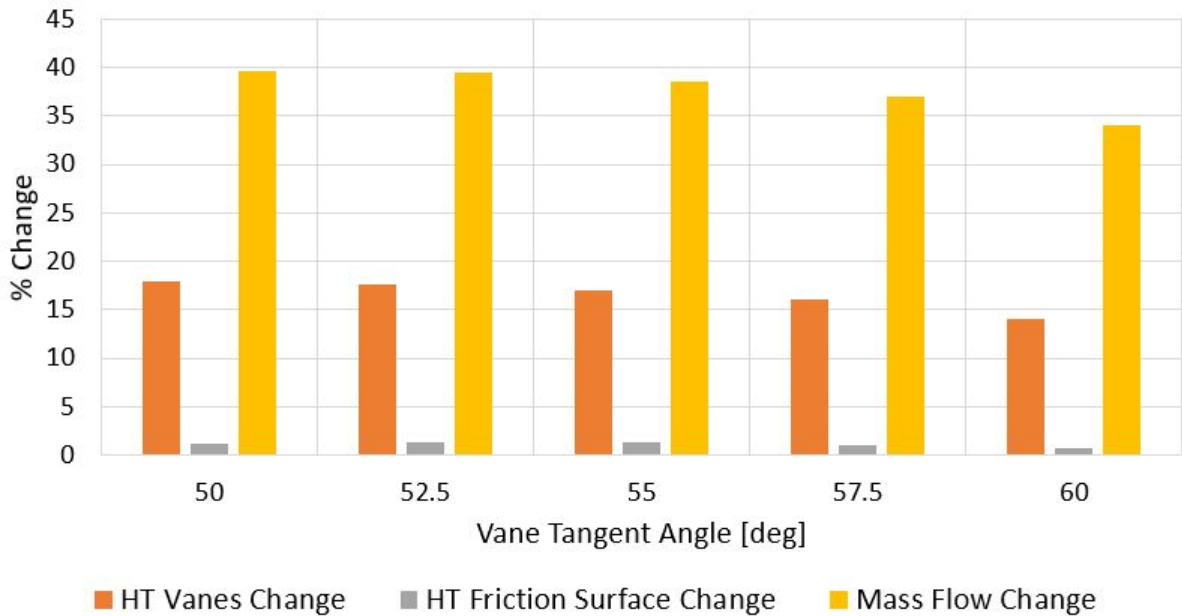


Figure 4.29: Influence of change of rotation direction on the cooling performance in tangential vanes.

It can be observed, from Figure 4.29, that there is a 40% reduction in the mass flow when the rotation direction is reversed in a tangential vane design and up to 10% reduction in the case of curved vane design, for the investigated designs. This can be one of the contributing factors to why there is a decrease in the heat transfer as the rotation direction is reversed. To investigate further, plots of velocity magnitude, skin friction coefficient

and the pressure coefficient are compared on the vanes, as can be seen in Figures 4.30 to 4.32.

The decreased cooling performance for the clockwise and anti-clockwise rotation direction for tangential vanes, may also be due to a lower flow interaction with the walls. To investigate this, the skin friction was studied on the vanes, as displayed in Figure 4.30.

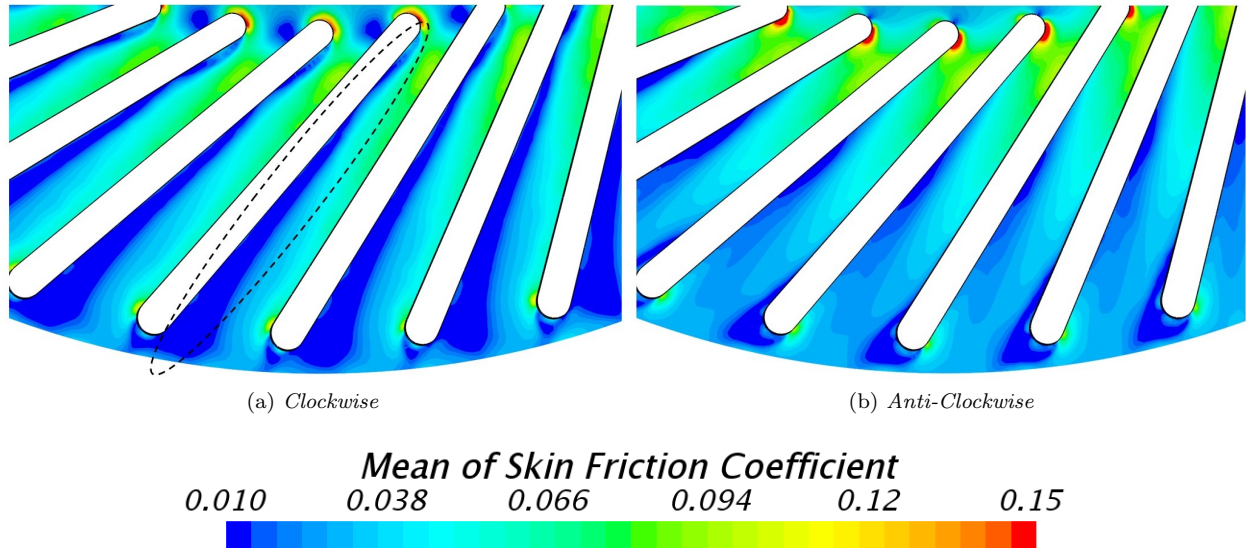


Figure 4.30: Influence of change of rotation direction on the skin friction coefficient in tangential vane design.

As can be seen, there are many large low skin friction area for the clockwise rotation direction on the left side of the vanes. This can be due to that the flow does not interact well with the walls, owing to lower heat transfer in the case of clockwise rotation.

Correlating the area of low skin friction to a contour of the velocity magnitude with streamlines, Figure 4.31, shows that there is re-circulation in the case of clockwise rotation while the flow stays more attached in the case of anti-clockwise rotation. Once more, this indicates that the flow interacts less with the wall, and even though the velocity is higher in part of the vane, less heat is transferred.

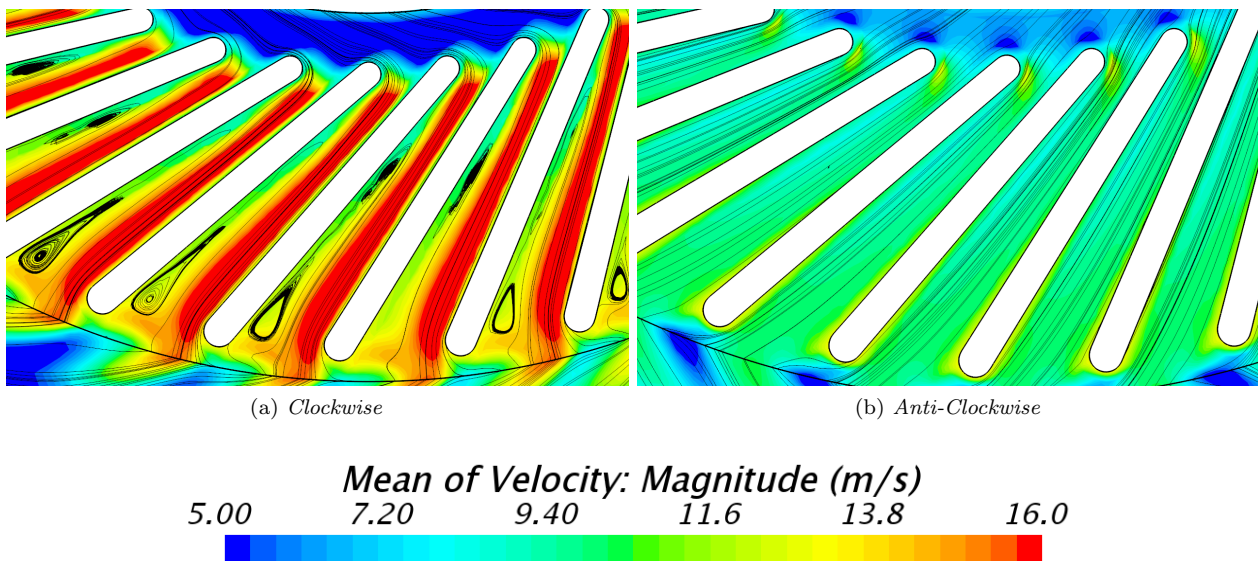


Figure 4.31: Influence of change of rotation direction on the velocity magnitude in a tangential vane design.

It is also interesting to note that there is high pressure region near the entry of the vanes in the clockwise rotation direction, as shown in the pressure coefficient contour in Figure 4.32, creating an effect that is similar to a 'blockage' effect, which results in further decreased mass flow in the clockwise direction.

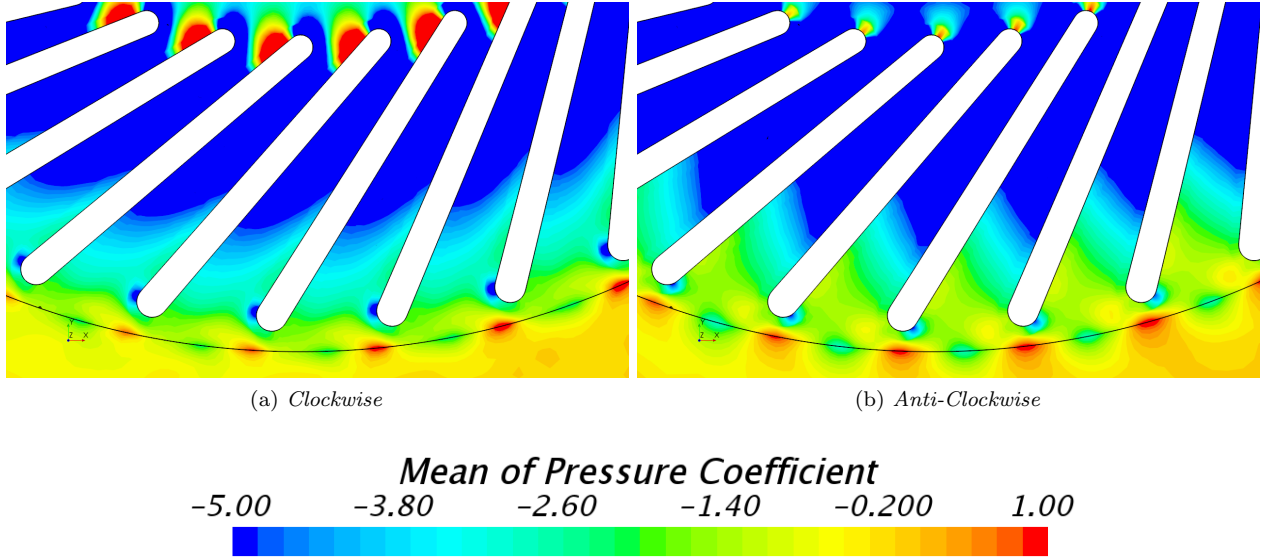


Figure 4.32: Influence of change of rotation direction on the pressure coefficient in tangential vane design.

Combining these effects, i.e. a less uniform flow, lower interaction with the walls, a higher pressure at the inlet does not make it come as a surprise that the heat transfer is decreased.

In the theory section, it was stated that both the curved and tangential vane type discs are directional discs, matching with the findings presented here. In reality, a manufacturer commonly manufactures only one type of disc for both left and right wheel. So with this investigation, it can be concluded that the cooling performance of both the tangential type discs are highly rotation direction dependent, which means that in a production vehicle, the disc in one side performs effectively while the disc in the other side performs poorly. Similar effect can also be expected in a curved vane type, however, analysis have not been done in this work. Hence, manufacturers need to think of the trade-off between manufacturing cost, manufacturing complexity and the cooling performance before making a choice of the type of disc for a vehicle.

4.6 Adjacent Parts

So far, the simulations performed have only included the brake disc. This works well for investigation of the simulation methodology and performing comparisons of parameter variations, but does not capture the actual flow field of a vehicle brake disc since a brake disc is commonly surrounded by many adjacent parts, e.g. the calliper and brake shield. To investigate the flow field and better understand how the convective heat transfer performance changes based on adjacent parts, addition of the brake calliper and brake shield were compared to the baseline configuration, which only includes the brake disc. The decreased heat transfer and mass flow due to adding the adjacent parts is presented in Figure 4.33.

As can be seen, the effect of adding the brake shield mainly affects the friction surfaces whilst the addition of a calliper shows a big impact on both the performance of the vanes and the friction surfaces. As expected, the combination of adding both the brake shield and the calliper results in an even lower heat transfer performance.

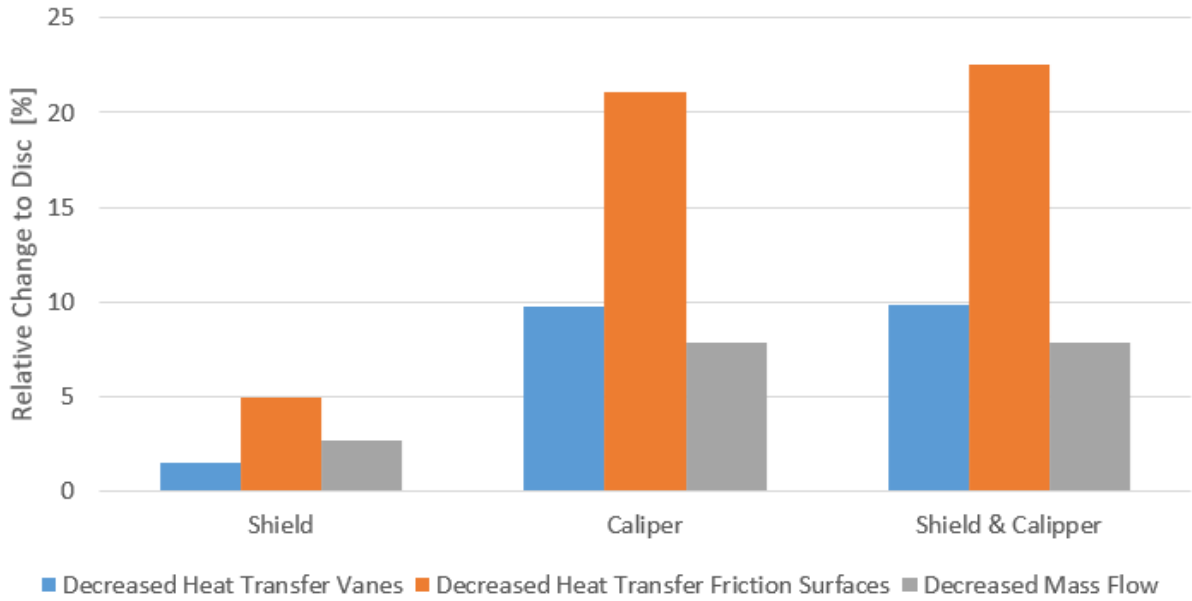


Figure 4.33: *Effect of additional parts on heat transfer and mass flow.*

To investigate why the heat transfer performance is decreased when a brake shield is added, it is first examined where on the brake disc the heat transfer is decreased. Figure 4.34 showcases the Specified Temperature HTC distribution on the inner friction surface, i.e. the one covered by the brake shield.

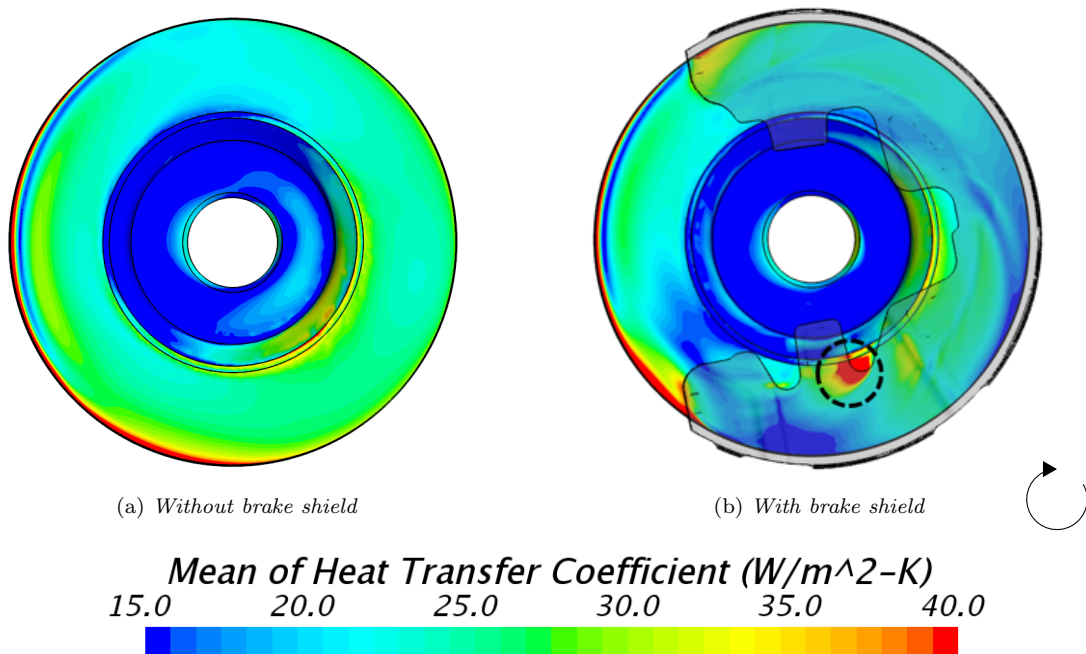


Figure 4.34: *Specified Temperature HTC on the inner friction surface. Oncoming flow from the left.*

As can be seen, the overall distribution of the Specified Temperature HTC is lower in the areas blocked by the dust shield except for one area. On the contrary, the other side of the brake disc shows very similar HTC distributions, indicating that the 5% heat transfer decrease for the friction surfaces comes from the decrease of the performance of the inner friction surface. To further investigate the area where the HTC is increased, velocity vectors are plotted in that area, Figure 4.35

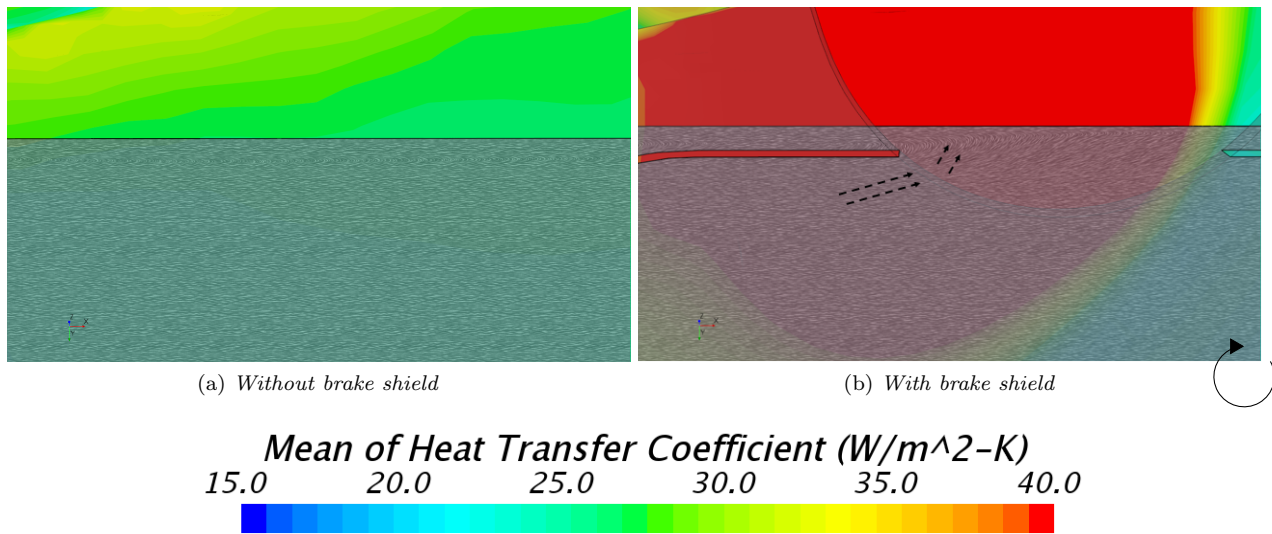


Figure 4.35: Close-up view of the high HTC area in Figure 4.34. Specified Temperature HTC on the friction surface and Vectors indicating the flow direction on a cut-plane.

From the figure, it can be observed that the flow travels almost normal onto the disc, instead of travelling parallel to the disc, which is the case when the brake shield is not present. Another observation is that the velocity next to the disc, in the high HTC area, is higher when the brake shield is present. These effects are similar to those of a diffuser, and are likely occurring due to the shape of the brake shield. The result of this is that more flow interacts with the brake disc in this area, whereby more heat can be transferred, whereby a higher HTC is obtained. However, this reasoning only applies in this area, and as seen from Figure 4.34, the heat transfer from the disc is decreased in most other areas. The reason for the overall decreased heat transfer performance is that the velocity and amount of flow interacting with the friction surface is decreased due to addition of the brake shield. Nonetheless, the finding of increased HTC in one area indicates that with an innovative design of the brake shield, the performance of the brake disc can likely be increased by the addition of the brake shield.

According to Figure 4.33, adding the brake calliper has a big impact on the performance of the disc. Once more, to find out how the calliper affects the heat transfer distribution, the Specified Temperature HTC contours are investigated, this time on the outside friction surface of the brake disc, Figure 4.36

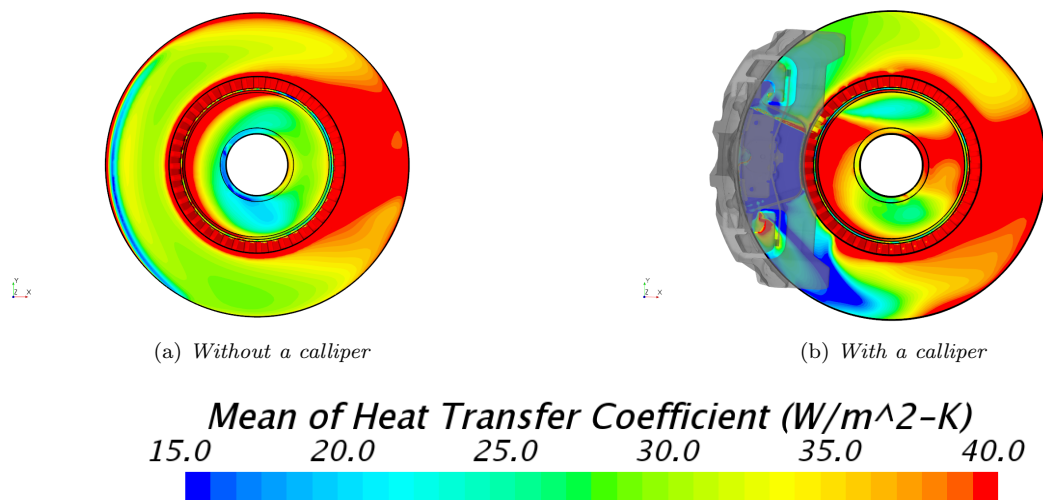


Figure 4.36: Specified Temperature HTC on the outer friction surface.

As can be seen in Figure 4.36, adding the brake calliper results in a predicted Specified Temperature HTC of 0 in the friction surface to pad contact area. In reality, this is of course not the case since the disc would physically rotate, whereby the complete friction surface would dissipate heat due to it not being stationary. Although, it is expected for the area to have a low HTC prediction compared to the rest of the disc due to less interaction with the air because of the obstruction of the brake pads. When simulated in steady-state with MRF, i.e. the disc not actually being rotated, and the brake pads of the calliper are located on the friction surface, no heat transfer is permitted from the friction surface at the pad location. This means that the heat transfer decrease of 20% from the friction surfaces does not portray the real picture. To gain a quick estimation of the heat transfer from the surface covered by the pad for a rotating the disc, the percentage area of the pad in relation to the friction surface can be multiplied with the heat transfer from the friction surface, as seen in equation 4.3.

$$Q = \frac{A_{pad}}{A_{FrictionSurface}} Q_{simulation} = 113W \quad (4.3)$$

This is only valid under the assumption of that the heat transfer will scale linearly. An increase of 113 W per friction surface would result in a significantly less decreased heat transfer performance. Instead of a 20% decrease, the decrease of the performance of the friction surfaces would be only 6.5%. This 6.5% can be explained by the same reason as for introducing the brake shield, i.e. that the calliper obstructs and slows the interacting flow to and from the friction surfaces.

As presented in Figure 4.33, the addition of the brake calliper also decreased the performance of the vanes by 10% and mass flow by 8%. To explain why, the pressure distribution in the vanes is investigated using the coefficient of pressure as seen in Figure 4.37

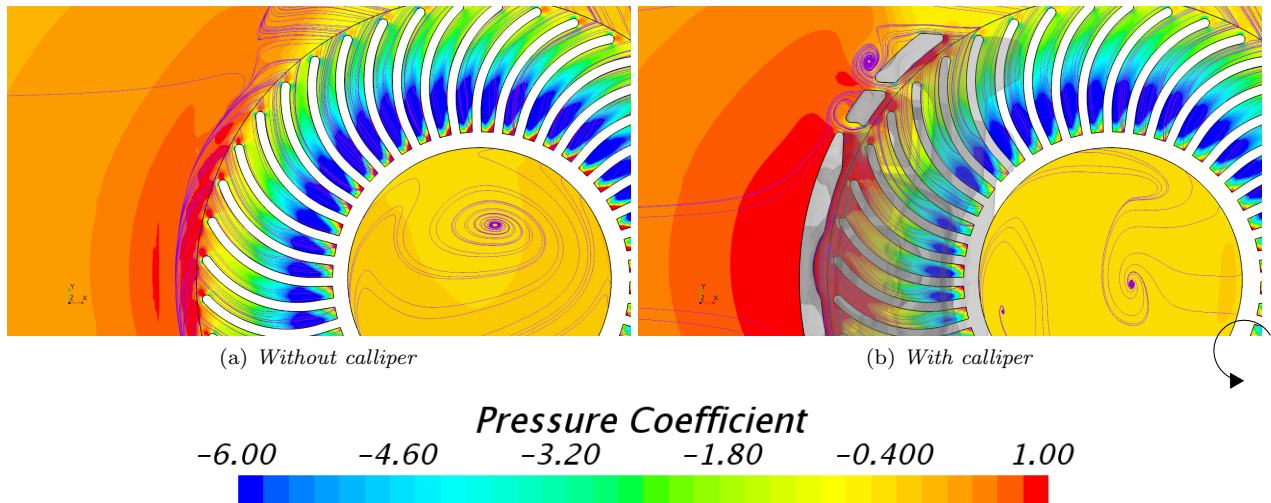


Figure 4.37: Coefficient of pressure inside of the vanes.

As can be seen from Figure 4.37, the calliper will halt the oncoming flow, resulting in a larger stagnation area. Additionally, a stagnation area is obtained on the inside of the calliper. Ideally, this would be a low pressure region to aid the pumping effect of the brake disc. However, due to the high pressure region, it will instead decrease the pumping effect through the vanes nearby the calliper. Note that it does not only affect the pressure at the outlet of the vanes, but instead affects the whole pressure distribution in the vanes for the worse, meaning that it will decrease the mass flow through these vanes. The effect of this on the heat transfer can be seen in Figure 4.38, where the Specified Temperature HTC is lower in the vanes covered by the calliper.

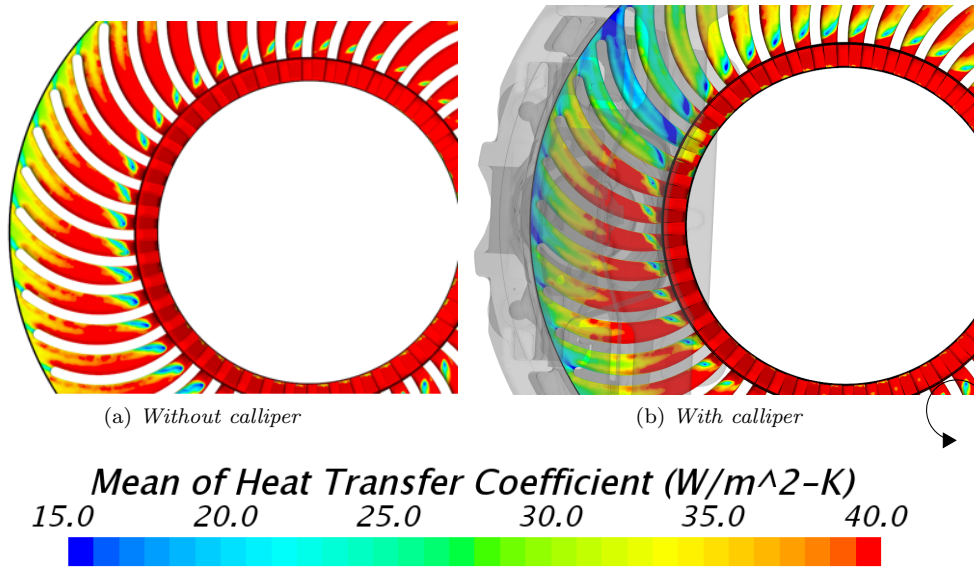


Figure 4.38: Specified Temperature HTC on the vanes.

Unsurprisingly, adding both the calliper and brake shield decreased the heat transfer even further. Once more, it is important to note that the decrease is likely over predicted due to that the pad-disc contact area will not transfer heat. As can be seen in Figure 4.33, adding both the shield and calliper further decreases the heat transfer from the friction surface. This is likely due to that more of the friction surface is being covered compared to only having the calliper or brake shield present. In turn, the flow will be further obstructed, resulting in less flow interacting with the brake disc whereby less heat can be dissipated. For the vanes, the heat transfer is not further decreased when the brake shield and calliper are added compared to only simulating the calliper, likely being due to that the brake shield only has a minimal effect on the vanes due to it not obstructing the vanes, nor interacting to much with the air travelling through the vanes. Another observation is that the cumulative heat transfer from the vanes and friction surfaces is not equal to the sum of the heat transfer decreases. One reason that can be found from studying the Specified Temperature HTC contours is that on the inner friction surface, i.e. the one covered by the brake shield, both the calliper and brake shield decreased the heat transfer separately. A few areas had very low heat transfer, and since the heat transfer does not become negative the accumulative effect will not be the same as the summed effect.

4.7 Gravity

For some conditions, gravity may play a big role, i.e. in natural convection, and in others it may only play a small role, i.e. in forced convection. To investigate when and how gravity influences brake cooling performance, simulations run with gravity were compared to simulations run without gravity for scenarios where natural convection and forced convection are expected to occur.

Unsurprisingly, for a standing still disc, i.e. a rotation velocity of 0 rpm, the addition of gravity to the simulation increases the heat transfer by almost 50 times. The reason being that without gravity, it would not be possible to capture the effect of the density changes, i.e. the buoyancy forces, meaning that a simulation for a non-rotating disc without gravity would be an invalid simulation since it is not realistic.

The effect of gravity was also investigated for a rotating disc at 100, 400, 800 and 1200 rpm. Comparing the heat transfer from the disc between simulations with gravity and without gravity yielded the result of a 2 % difference in the heat transferred from the disc. Additionally, the effect of running with and without gravity on computational time, was also investigated and found to be insignificant. Since the effect of gravity on simulation run-time is low, it can be included in the model without further consideration, even though it has been shown to not affect forced convection flows much.

4.8 Radiation

The results presented up until now have only been simulating convective heat transfer. In reality, the total heat transfer would be higher due to the influence of radiation and conduction. Radiation is dependent on the temperature as explained in Section 2.4.3 equation 2.12, i.e. dependent on the temperature to the power of four. To verify the applied Star-CCM+ model for radiation, equation 2.12 has been applied, with the same emissivity as in Star-CCM+, to estimate the radiation HT from the friction surface and added to the resulting graph, Figure 4.39.

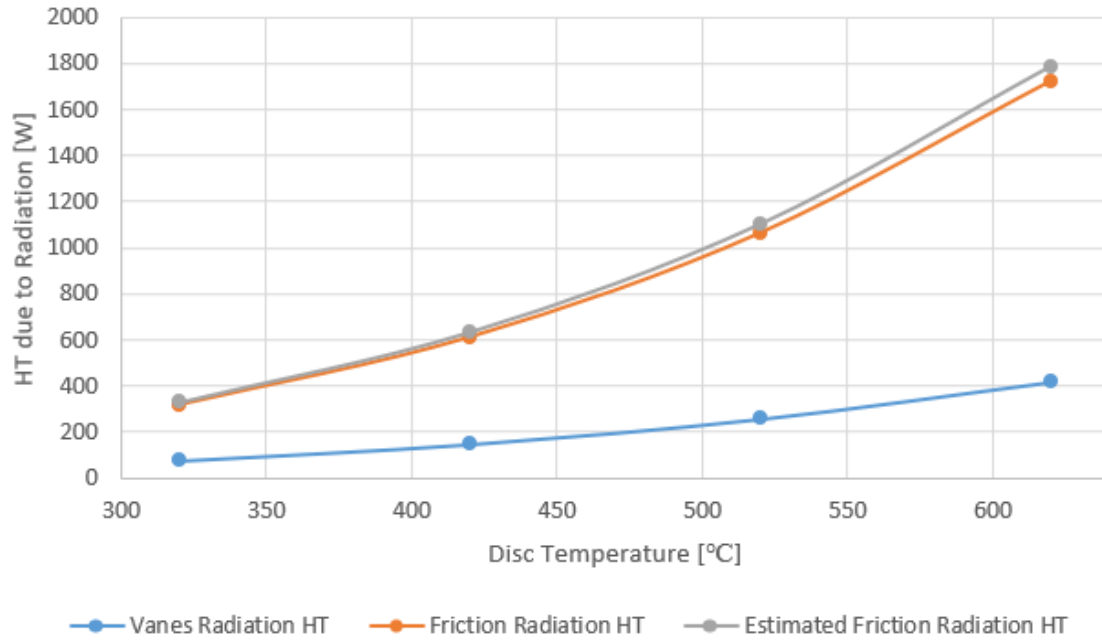


Figure 4.39: Heat transfer due to radiation for a disc temperature ranging from 320 to 620 °C.

An observation from Figure 4.39 is that the estimated radiation heat transfer from Star-CCM+ correlate well with equation 2.12 for the friction surfaces, visualized by the blue and grey line in the figure. One can also note that the radiation heat transfer was considerably higher for the friction surfaces compared to the vanes. The reason for this is that the exterior surfaces of the disc, i.e. the friction surfaces, can radiate freely out into the surrounding medium. On the interior surfaces, i.e. the vanes, some of the radiated heat will again be absorbed by the vanes. The temperature in the vanes will be higher compared to the surrounding medium temperature around the disc, i.e. T_2 in equation 2.12 will be higher, thus decreasing the radiation heat transfer as dictated by equation 2.12.

To understand the effect of radiation compared to only simulating convective heat transfer, the percentage heat transfer addition due to radiation was studied at different temperatures. The percentage increase compared to a convective heat transfer case at a rotational velocity of 800 rpm (100km/h) is displayed in Figure 4.40.

As can be seen, the role of radiation becomes increasingly important as the temperature increases due to the power of four dependency on temperature. However, during normal driving conditions, since the disc mainly does not operate at high temperatures, radiation plays a minor role. Additionally, the computational time was monitored, which showed that the simulation run-time increased by an average of 90 % when the surface-to-surface radiation model was added. Another consideration to keep in mind is that the simulations were performed without adjacent parts. This results in that the exterior surfaces of the disc can radiate freely, whilst in the car the adjacent parts would get heated and hence, the radiation from the brake disc would be lower.

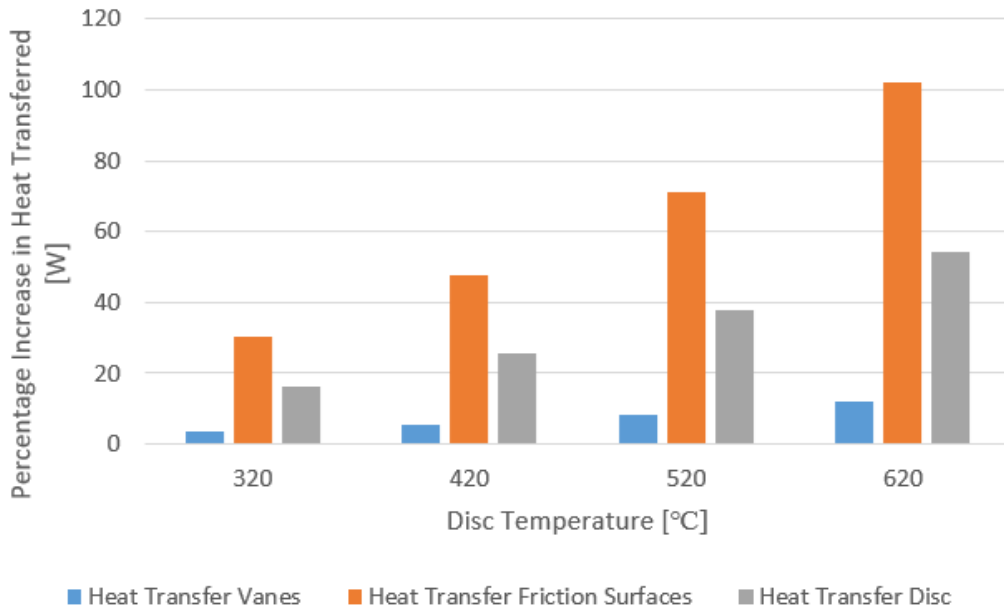


Figure 4.40: Heat transfer increase due to radiation for a disc temperature ranging from 320 to 620 °C.

4.9 Solid Mesh Dependency Investigation

Similar to the mesh independent solution for the fluid domain, a mesh independent solution for the solid domain was found. For this purpose, 6 different meshes were generated, ranging from 7000-300,000 cells, and run in steady-state according to the simulation methodology presented in Section 3.2.4. Based on the converged simulations, the heat transfer from the vanes and the required computational run-time is plotted against the number of cells, as displayed in Figure 4.41.

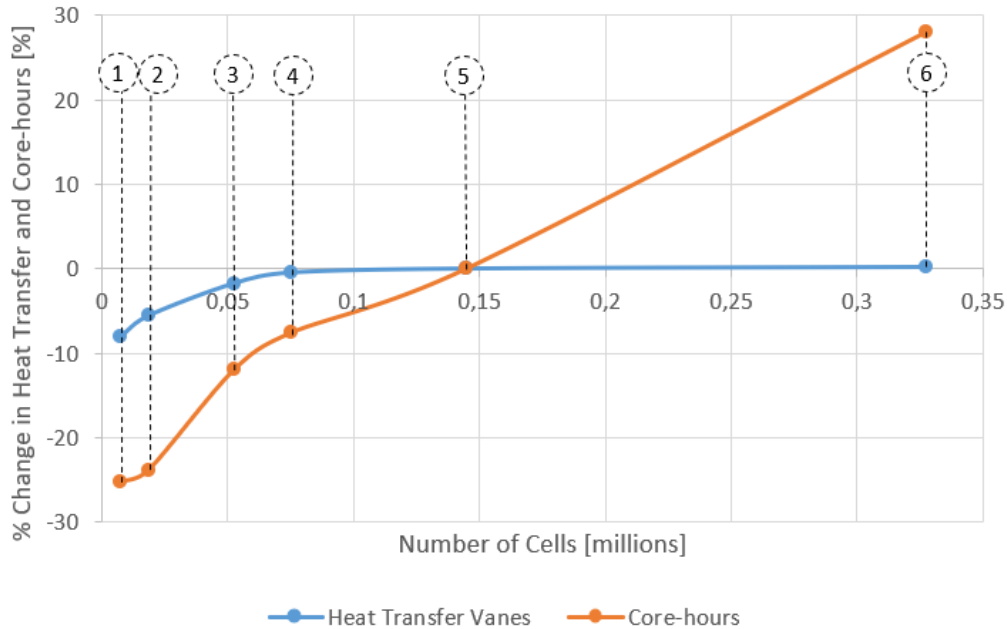


Figure 4.41: Computational time and heat transfer from the vanes as a function of number of cells.

It can be observed that, already at 75 000 cells, i.e. case 4, the heat transfer from the vanes of the disc stabilize. The reason for only requiring 75 000 cells, compared to e.g. 400 000 cells for the vane region, is that the gradients are much lower compared to for a fluid region.

4.10 Cool-down Simulations

Most simulations performed so far have been simulated in steady-state. Simulating in steady-state means that the time required for e.g. cooling the brake disc from a certain time cannot be captured. In order to do so, the simulation has to be run as a unsteady simulation. As shown in Section 4.1.2, the time required for a unsteady simulation with a fluid domain can be 10 times as high compared to a steady-state simulation. To keep the computational time low, it was chosen to only simulate the solid region since the solid region does not require an as low time-step as a fluid region. Simulating with only a solid region will require a convection boundary condition and a HTC to be specified for the boundaries. By using the functions defined in Section 4.2.4, Figure 4.11, a cool-down simulation of a heated brake system at 400, 800 and 1200 rpm, equaling approximately 50, 100 and 150 kph respectively, could be performed. Utilizing these HTC functions implies that the HTC for the brake disc can be averaged and assumed the same for all surfaces, since the curve is applied to all boundaries, but does not assume the HTC to be linear as a function of temperature. From these simulations, three cool-down curves could be created, as displayed in Figure 4.42.

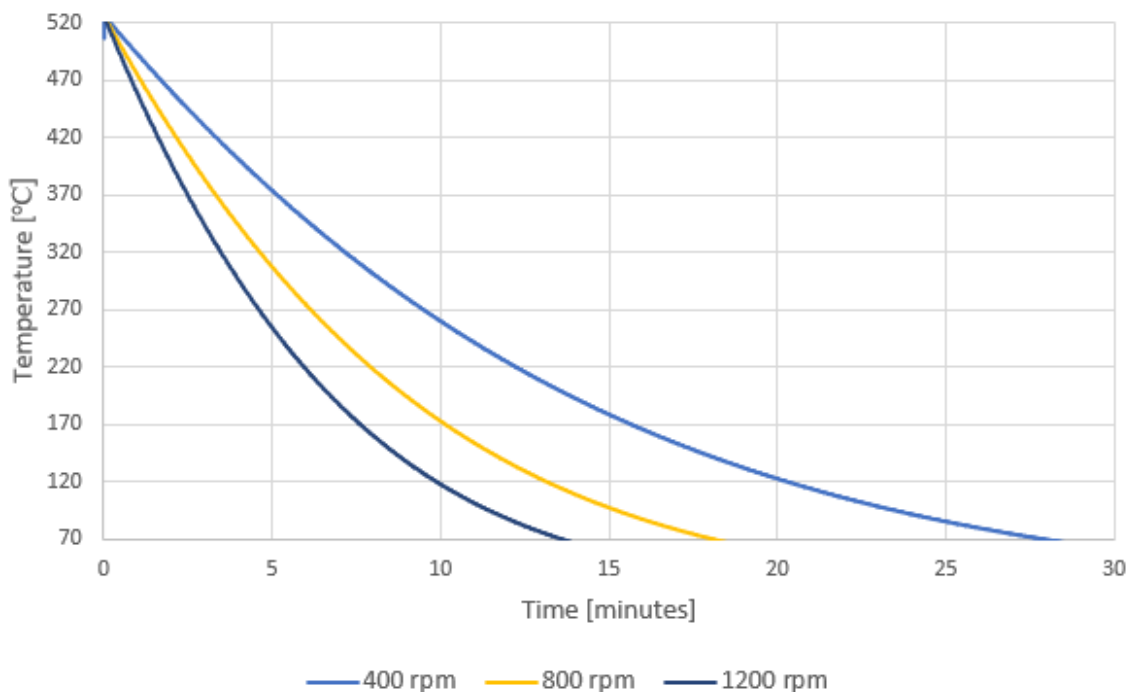


Figure 4.42: The time required to cool-down a brake disc from 520 °C to 20 °C for three different rotational velocities.

As can be seen, decreasing the rotational velocity from 1200 to 400 rpm, results in a twice as long cool-down time to reach 100 °C. In papers with similar cool-down studies at constant velocity, the cool-down time was found to be 365 seconds cooling down from 600-100 °C at 140 kph [29]. However, the model in [29], includes radiation and conduction, which can be one of many explanations why their cool-down time was lower compared to the findings in Figure 4.42. Another study showed a cool-down time of 25 minutes for a constant velocity cool-down from 450-50 °C [6]. The obtained results are in a similar range, indicating that this method can be used to predict cool-down results. However, further investigations and verification would be needed to see how accurate it is in terms of the exact estimation of the time required for cooling the brake disc.

Since the method only simulates the solid region, the time-step can be specified as 1 second, instead of 0.21 milliseconds which would equal 1 degree per time-step at 800 rpm, and would be needed for a fluid region. Even though, if a semi-unsteady approach, solving the energy and flow part individually, would have been applied, it would require more time than only simulating the solid region. Thereby there is a huge time-reduction with this methodology, even though it requires 5-7 steady-state simulations to approximate the HTC as a function of temperature. And after having solved the function for the HTC, it can be scaled by simulating only 2-3

more simulation, reducing the required computational time further as explained in Section 4.2.4. Considering the low computational time required for running cool-down simulations according to this methodology would make it very valuable if the method is accurate.

5 Conclusions

Approaches to evaluate cooling performance of brake discs were performed using CFD simulations with MW, MRF and RBM, in order to simulate the movement of air in the disc due to rotation. It was found that MW required the least computational time and as expected, did not provide realistic results due to it only applying a boundary condition to the walls and no condition to the air region. RBM required significantly greater computational time, but was more realistic and good for detailed analysis. MRF showed comparable results with RBM for aerodynamic and thermodynamic properties and converged for a lower computational time. Hence, for overall predictions of the heat transfer or mass flow through a brake disc, MRF can be applied to compute sufficiently accurate results at a low computational cost. With respect to turbulence models, $k - \varepsilon$ model showed fairly good results compared to literature for the steady case and considering the computational time and also giving a fairly good estimation of the results, URANS ($k - \varepsilon$ model) was used in the unsteady case.

From the investigation of the four field function definitions for the HTC, it was found that the Virtual Local and Local HTC are first cell height dependent, whilst the Specified Temperature and Specified $y+$ HTC are not. Furthermore, the reference temperature of the Specified $y+$ HTC was found to be velocity dependent and thus, comparing two simulations with different velocities, would result in different reference temperatures. Hence, the Specified Temperature HTC provides a better way to evaluate and compare the cooling performance of brake discs due to it being applicable in both low and high $y+$ regions, as it is not first cell height dependent, as well as that the reference temperature is not velocity dependent.

To investigate ways of saving computational time, the Virtual Local HTC and the possibility of linearizing the Specified Temperature HTC as a function of rotational velocity and temperature were investigated. Both strategies reduce the computational time since the Virtual Local HTC can predict the heat transfer without solving the energy equation and linearizing the Specified Temperature HTC results in less simulations to be run. It was found that the Virtual Local HTC can be applied, and correlates fairly well with the Specified Temperature HTC for a varying rotational velocity. When the measure was applied on different brake disc geometries, it predicted an twice as high HTC compared to the Specified Temperature HTC. This indicates that the Virtual Local HTC can only be applied for early indications to evaluate what is better or worse, not quantifying how much better or worse. The linearization of the Specified Temperature HTC as a function of rotational velocity and temperature correlated well in a close range to from where the linearization was performed. However, especially for the linearization based on temperature, moving further away (± 400 °C) the linearization showed a deviation of 10 %, clearly indicating that it is not fully linearizable. In addition to this, it was also investigated if a non-linearly estimated HTC function of temperature could be scaled to different rotational velocities, which was found to be functioning.

Factors affecting the disc heat transfer performance were studied and the parameters which improved the heat transfer performance of the disc are: increased free stream velocity, increased rotational velocity, increased disc temperature and decreased ambient temperature. The main reason being, that the temperature gradient and mass flow are altered as the parameters are changed.

The performance of three vane types, i.e. radial, tangential and curved vanes, was investigated. The disc vane geometric parameters, i.e. the vane thickness, vane tangent and vane curvature, play a major role in the cooling performance. Increasing vane width, allows more mass flow through the vanes and hence increases the cooling performance. However, a too wide vane leads to flow separation in the vanes, thereby decreased the cooling performance. For the vane tangent, the cooling performance is first improved as the mass flow is increased by increasing the vane tangent due to a more favorable inlet angle, but starts to degrade after a saturation point as the vane inlet width is decreased, thereby restricting the mass flow. Modifying the vane curvature showed a similar trend in the case of a curved vane type. The tangential and curved vane types are rotation direction dependent, i.e. they perform better in one direction but not in the opposite direction. Radial vane brake discs are not influenced by changing the direction. Hence, a wise choice has to be made considering the trade-off between the cooling performance and the manufacturing cost and complexity.

By adding brake system parts, such as the dust shield and brake caliper, the cooling performance is decreased.

The dust shield mainly influences the convective heat transfer from the inner friction surface, whilst the caliper decreases the cooling performance primarily by blocking the oncoming air into and onto the disc. One area showed an increased heat transfer with the brake shield due to it increasing the amount of interacting flow with the brake disc. Hence, the cooling performance of the brake disc can likely be improved with an innovative brake shield.

The effect of gravity was shown to be negligible when forced convection was dominating, as well as that the effect on computational time was insignificant. In order to simulate natural convection, gravity has to be included. Hence, gravity can be included into the simulations since it had an insignificant effect on the computational time, but may be of importance to capture the heat transfer for natural and mixed convection cases.

The effect of radiation is important at high temperature and has shown to increase the heat transfer, for this type of brake disc, by 50 % at a temperature of 600°C. At low temperature, e.g. 300 °C, the effect of radiation is low, especially on the vanes, but still has an impact of the heat transfer from the friction surfaces. However, it should be kept in mind that the radiation actually would be lower due to that the adjacent parts will have a higher temperature than the ambient temperature, and hence the effect of radiation will be smaller. The computational cost of including the radiation model was found to be around 90 % of additional run-time. It was also shown that the radiation heat transfer from the friction surfaces computed by the surface-to-surface radiation model correlates well with equation 2.12. If the total heat transfer should be captured accurately, the radiation model should be included. The radiation heat transfer is especially important to consider at high temperatures, but at low temperatures it can be neglected, due to its dependency on temperature to the power of four.

For the unsteady cool-down simulations, only the solid region was considered. This resulted in being able to achieve cool-down curves for a significantly decreased computational time due to being able to utilize a higher time-step. The cool-down time from 520°C to 120°C was found to be approximately 20, 13 and 10 minutes for a rotational velocity of 400, 800 and 1200 rpm respectively. Even though the approach requires several steady-state simulations, the computational run-time is still lower. The method looks to be predicting promising results for a low computational time, but would require comparison to experimental values or a fully unsteady simulation to answer how accurate it is.

5.1 Future Scope

Simulations can provide good estimate of the performance behaviour and help to reduce the amount of physical test time. In this work, many different areas that influence the cooling performance have been investigated, but there is always scope for further developments and new ways to approach a problem. Having said that, studies involving other turbulence models and more unsteady simulations can be performed to investigate ways to better replicate reality.

Additionally, the parameter variations performed were investigated independent of each other to find trends while running as few simulations as possible. Commonly, the parameters are dependent on each other, i.e. if the car increases its speed, the brake disc rotational velocity will increase as well as that the oncoming velocity will likely increase. Thus, it can be valuable to perform a design of experiments of the parameter variations.

A delimitation of this work, is that degrading of brake components throughout the life-cycle is not considered. This, as well as the brake disc material, can have a big effect on the heat transfer performance and can yield different results than those achieved in this work, which is why it is important to perform further investigations if the material properties are altered.

Another area where additional investigations can be performed is in the field of brake disc designs. Similar to the parameter variations, the parameters were changed independent of each other to keep the number of simulations to a minimum. This leaves room for improvement, as well as that vane patterns such as pillar, diamond and mixed vane pattern designs can be investigated. If the weight is considered into this investigation, it would provide even more beneficial results.

For the adjacent parts, only the caliper and the brake shield were investigated. Obviously, the vehicle wheel, wheel house, cooling ducts, etc. will have a big effect on the heat transfer and flow field around the brake disc which is why it would be of value to add such parts. Another consideration is that the additional part simulations were investigated for convective heat transfer only, meaning that the results might be different when including radiation and conduction. Hence, it can be of importance to continue the investigation of adjacent parts with more parts and models.

Finally, the concluded method for simulating unsteady cool-down simulations, with only simulating the solid region using convection boundary conditions and a non-linear HTC function, showed promising results. However, to be sure of how accurate it is, the method needs to be verified with either real test-data or a fully unsteady simulation including both the solid and fluid region.

References

- [1] E. Palmer et al. "Analysis of Air Flow and Heat Dissipation from a High Performance GT Car Front Brake". *SAE World Congress Exhibition*. SAE International, 2008. DOI: <https://doi.org/10.4271/2008-01-0820>. URL: <https://doi.org/10.4271/2008-01-0820>.
- [2] S. Sun et al. "A Coupled Approach to Truck Drum Brake Cooling". *SAE 2015 Commercial Vehicle Engineering Congress*. SAE International, Sept. 2015. DOI: <https://doi.org/10.4271/2015-01-2901>. URL: <https://doi.org/10.4271/2015-01-2901>.
- [3] D. Mukutmoni et al. Role of Accurate Numerical Simulation of Brake Cooldown in Brake Design Process. *SAE International Journal of Passenger Cars - Mechanical Systems* **5.4** (2012), 1199–1210. ISSN: 1946-4002. DOI: <https://doi.org/10.4271/2012-01-1811>. URL: <https://doi.org/10.4271/2012-01-1811>.
- [4] D. J. W. M. Streitenberger Dr. S. Yigit. Multiphysical Simulation of Brake Fluid Temperature during Mountain Descent. *Fortschritt-Berichte VDI Nr. 806* (Oct. 2017), 97–114.
- [5] J. Happian-Smith. *An introduction to modern vehicle design*. Butterworth-Heinemann, Oxford, 2001. ISBN: 0080523048, 9780080523040, 0750650443, 9780750650441.
- [6] S. Yigit et al. A Monolithic Approach to Simulate the Cooling Behavior of Disk Brakes. *SAE International Journal of Passenger Cars - Mechanical Systems* **6.3** (2013), 1430–1437. DOI: <https://doi.org/10.4271/2013-01-2046>. URL: <https://doi.org/10.4271/2013-01-2046>.
- [7] T. Schuetz. "Cooling Analysis of a Passenger Car Disk Brake". *SAE 2009 Brake Colloquium and Exhibition*. SAE International, 2009. DOI: <https://doi.org/10.4271/2009-01-3049>. URL: <https://doi.org/10.4271/2009-01-3049>.
- [8] R. Krüsemann and G. Schmidt. "Analysis and Optimization of Disk Brake Cooling via Computational Fluid Dynamics". *International Congress Exposition*. SAE International, 1995. DOI: <https://doi.org/10.4271/950791>. URL: <https://doi.org/10.4271/950791>.
- [9] D Chan and G. W. Stachowiak. Review of automotive brake friction materials. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* **218.9** (2004), 953–966. DOI: 10.1243/0954407041856773. eprint: <https://doi.org/10.1243/0954407041856773>. URL: <https://doi.org/10.1243/0954407041856773>.
- [10] S. M. Reddy, J. M. Mallikarjuna, and V. Ganesan. "Flow and Heat Transfer Analysis of a Ventilated Disc Brake Rotor Using CFD". *SAE World Congress Exhibition*. SAE International, 2008. DOI: <https://doi.org/10.4271/2008-01-0822>. URL: <https://doi.org/10.4271/2008-01-0822>.
- [11] R. B. Stephen John Ruiz. Ventilated Brake Rotor (2008).
- [12] B. S.p.A. The right Brembo brake discs to choose to get the best braking performance from the Opel Corsa E (2018). URL: <http://www.brembo.com/en/company/news/the-right-brembo-brake-discs-to-choose-to-get-the-best-braking-performance-from-the-opel-corsa-e>.
- [13] D. Yogesh Satish. *Effect of Design Parameters on Thermal Performance of a Vane Type Disc Brake Rotor*. Master Thesis, Western Michigan University, 2017.
- [14] D. P. and D. G. MacManus. "Aerodynamic investigations of ventilated brake disks". *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*. 2005.
- [15] Z. Chi, Y. He, and G. Naterer. Convective Heat Transfer Optimization of Automotive Brake Discs. *SAE International Journal of Passenger Cars - Mechanical Systems* **2.1** (2009), 961–969. DOI: <https://doi.org/10.4271/2009-01-0859>. URL: <https://doi.org/10.4271/2009-01-0859>.
- [16] W.-T.W.S.F.G. X. Hongbin Yan. Role of vane configuration on the heat dissipation performance of ventilated brake discs. *Applied Thermal Engineering* **136** (2018), 118–130. DOI: <https://doi.org/10.1016/j.applthermaleng.2018.03.002>.
- [17] P. N. Amrish. Computer Aided Design and Analysis of Disc Brake Rotors. *Advances in Automobile Engineering* **5.2** (2016). DOI: <https://doi.org/10.4172/2167-7670.1000144>. URL: <https://doi.org/10.4172/2167-7670.1000144>.
- [18] Andersson, Bengt and Chalmers University of Technology and Department of Chemical and Biological Engineering, Chemical Reaction Engineering. Computational Fluid Dynamics for Engineers (2012).
- [19] R. Schwarze and F. Obermeier. Performance and limitations of the unsteady RANS approach. *Pamm* **6.1** (2006), 543–544.
- [20] T. Hobeika. "Wheel modelling and cooling flow effects on car aerodynamics". English. PhD thesis. 2018. ISBN: 9789175976877; 9175976870;

- [21] M. J. Ciolfi and P. E. B. de Mello. "Heat Transfer Analysis in a Usual Ventilated Brake Disc". *12th SAE Brasil International Brake Colloquium Engineering Display*. SAE International, 2015. DOI: <https://doi.org/10.4271/2015-36-0019>. URL: <https://doi.org/10.4271/2015-36-0019>.
- [22] Incropera, F. P., DeWitt, D. P., Bergman, T. L., Lavine, A. S. *Fundamentals of Heat and Mass Transfer*. John Wiley Sons, 2006. ISBN: 0471457280, 978-0471457282.
- [23] E. Palmer and W. Jansen. "Development of a High Fidelity CAE Model for Predicting Brake System Temperatures". *WCXTM 17: SAE World Congress Experience*. SAE International, 2017. DOI: <https://doi.org/10.4271/2017-01-0145>. URL: <https://doi.org/10.4271/2017-01-0145>.
- [24] *Star CCM+ manual: Steve Portal*. Siemens Product Lifecycle Management Software Inc. URL: https://documentation.thesteveportal.plm.automation.siemens.com/starccmplus_latest_en/index.html?param=NzS0g&authLoc=https://thesteveportal.plm.automation.siemens.com/AuthoriseRedirect.
- [25] S. I. Inc. Choose an Appropriate Mesh Type, Release 16.2 (2018). URL: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/flu_ug/flu_ug_choicemeshtypes.html.
- [26] Symscape. Polyhedral, Tetrahedral, and Hexahedral Mesh Comparison (2018). URL: <https://www.symscape.com/polyhedral-tetrahedral-hexahedral-mesh-comparison>.
- [27] N. Kapas and A. Jayasundera. High-Fidelity Transient Thermal Modeling of a Brake Corner. *SAE International Journal of Passenger Cars - Mechanical Systems* **9.3** (2016), 1184–1198. ISSN: 1946-4002. DOI: <https://doi.org/10.4271/2016-01-1929>. URL: <https://doi.org/10.4271/2016-01-1929>.
- [28] B. Choi, J. Park, and M. Kim. "Development of the Virtual Test Technology for Evaluating Thermal Performance of Disc Brake". *SAE World Congress Exhibition*. SAE International, 2009. DOI: <https://doi.org/10.4271/2009-01-0857>. URL: <https://doi.org/10.4271/2009-01-0857>.
- [29] K. Bhambare, M. Haffey, and S. Jelic. "Brake Duty Cycle Simulation for Thermal Design of Vehicle Braking System". *11th SAE Brasil International Brake Colloquium Engineering Display*. SAE International, 2013. DOI: <https://doi.org/10.4271/2013-36-0015>. URL: <https://doi.org/10.4271/2013-36-0015>.
- [30] R. Limpert. "Cooling Analysis of Disc Brake Rotors". *National Truck Meeting and Exposition*. SAE International, 1975. DOI: <https://doi.org/10.4271/751014>. URL: <https://doi.org/10.4271/751014>.
- [31] B. Jeong et al. "Optimization of Cooling Air Duct and Dust Cover Shape for Brake Disc Best Cooling Performance". *SAE Brake Colloquium Exhibition - 32nd Annual*. SAE International, 2014. DOI: <https://doi.org/10.4271/2014-01-2519>. URL: <https://doi.org/10.4271/2014-01-2519>.